AUTHC

NASA Technical Memorandum 101961
ICOMP-89-1

Institute for Computational Mechanics in Propulsion (ICOMP)

(NASA-TM-101961) INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION (ICOMP) Annual Report No. 3, 1988 (NASA) 62 p
CSCL 12A
N89-21595
Unclas
G3/64 0200239

Third Annual Report—1988
produced by the ICOMP Steering Committee

March 1989

NASA

CONTENTS

	Page
INTRODUCTION	1
THE ICOMP STAFF OF VISITING RESEARCHERS	2
RESEARCH IN PROGRESS	3
REPORTS AND ABSTRACTS	31
SEMINARS	39
WORKSHOPS	53
OTHER ACTIVITIES	54

INSTITUTE FOR COMPUTATIONAL MECHANICS

IN PROPULSION (ICOMP)

THIRD ANNUAL REPORT

1988

SUMMARY

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated jointly by Case Western Reserve University and the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1988.

INTRODUCTION

The Institute for Computational Mechanics in Propulsion (ICOMP) is jointly operated at the NASA Lewis Research Center by Case Western Reserve University and NASA Lewis under a Space Act Agreement. ICOMP provides a means for researchers with experience and expertise to spend time in residence at Lewis performing research to improve computational capability in the many broad and interacting disciplines of interest in aerospace propulsion. The organization and operation of ICOMP have been described in ICOMP Report No. 87-8 (NASA TM-100225), "The Institute for Computational Mechanics in Propulsion (ICOMP), First Annual Report," Nov., 1987, 14 pages. The activities of 1987 are described in ICOMP Report No. 88-1 (NASA TM-100790), "The Institute for Computational Mechanics in Propulsion (ICOMP), Second Annual Report," March, 1988, 46 pages.

The scope of the ICOMP research program is: to advance the understanding of aerospace propulsion physical phenomena; to improve computer simulation of aerospace propulsion components; and to focus interdisciplinary computational research efforts. The specific areas of interest in computational research include: fluid mechanics for internal flow; structural mechanics and dynamics; multivariable control theory and applications; and chemistry and material science.

The report summarizes the activities at ICOMP during 1988. It lists the visiting researchers, their affiliations and time of visit followed by reports of RESEARCH IN PROGRESS, REPORTS AND ABSTRACTS, and SEMINARS presented. Two other important activities of 1988 are also discussed. These are the WORKSHOPS sponsored by ICOMP during the summer of 1988 and the selection of a Director for ICOMP.

THE ICOMP STAFF OF VISITING RESEARCHERS

The composition of the ICOMP staff during 1988 is shown in figure 1. Fifty researchers were in residence at Lewis for periods varying from a few days to a year. The number in residence on a weekly basis is shown in figure 2. This figure clearly shows the "high season" during the summer months when there was a peak of twenty-nine in residence. Figure 3 is a photograph of the ICOMP Steering Committee and the visiting researchers, taken at a reception in July, 1988. Figure 4 lists the universities or other institutions represented and the number of people from each. The figure lists forty-three organizations. Figure 5 shows the growth of ICOMP during its first three years in terms of staff size, organizations represented and technical output as measured by the numbers of seminars, reports and workshops. The next sections will describe the technical activities of the visiting researchers starting with reports of RESEARCH IN PROGRESS, followed by REPORTS AND ABSTRACTS, and finally, SEMINARS.

RESEARCH IN PROGRESS

Dare Afolabi, Purdue University

During my 8-week residence at ICOMP, the effect of mistuning on the aeroelastic stability and forced response of turbomachine blades in a multi-stage engine was investigated. A presentation was made at the ICOMP Workshop on Unsteady Phenomena in Turbomachinery (July 20, 1988) on recent developments in this area.

Present work on mistuning in a multi-stage engine is an extension of previous studies on single stage rotors. The impact of mistuning is difficult to predict, even for a single stage. In fact, the results of various investigators of the problem are often so different as to be in conflict. This is due to the structural instability inherent in blade mistuning, such that even the mildest perturbation in a system's parameters often leads to a radically different behavior. In order to clarify the problem, I developed a generic approach by treating the problem to be solved as a generalized topological mapping in a Banach space. The development naturally leads to considerations in the singularity of smooth maps and, eventually, to Catastrophe Theory. Based on the limited results I have already obtained, it is certain that Catastrophe Theory can help explain why a very small perturbation (mistuning) may lead to considerable -- and sometimes abrupt -- changes in the qualitative behavior of a vibrating linear elastic system.

Suresh K. Aggarwal, University of Illinois at Chicago

A computational study of the turbulent evaporating and combusting methanol spray has been conducted. The first set of results focusses on the comparison of three vaporization models for a dilute combusting spray. Three models considered are thin-skin, infinite-diffusion and diffusion-limit models. The physical configuration for this case involves the vaporization of a stream of methanol droplets in a turbulent, methane-fueled diffusion flame.

The computational model is then extended to a more realistic combusting case with full interaction between the phases, i.e., the effects of liquid-phase processes on the gas-phase properties, which were neglected in the first calculations, are also included. The physical model considers a pressure-atomized methanol injection into quiescent air. A conserved scalar approach is employed and the state relationships required for the combustion model are obtained by using a partial equilibrium approach. Other features of the physical model are that the Favre-averaged equations are employed for the continuous-phase variables, that the k-E-g turbulence closure model is used, and that the drop-turbulence interaction is considered via the stochastic separated flow (SSF) model. Detailed numerical results on the structure of the turbulent combusting spray jet and on the comparison of three vaporization models are obtained. These results will be discussed in a NASA technical report.

F. A. Akl, Ohio University

The research was concerned with vibration analysis in parallel microcomputing architecture. A particular objective is to study the computational characteristics associated with the mapping of the generalized eigensolution of linear elastic finite element models onto a parallel microcomputing architecture. The work completed so far includes: a binary tree architecture for the creation and assembly of the stiffness matrices of

finite element models which has been implemented on the transputer; a new data management scheme for fully and partially assembled degrees-of-freedom; and communication links between various levels of the binary tree to handle forward elimination processes. Continuing work includes: backward substitution in binary tree architecture; solution of the auxiliary eigenproblem; and communication links to handle backward substitution processes. Also, an attempt will be made to integrate this algorithm with the graphics software developed by G. K. Ellis, former ICOMP researcher.

Alvin Bayliss, Northwestern University and Eli Turkel, Tel Aviv University

Work was begun on adaptive procedures for pseudo-spectral methods. These methods are global approximations which are highly accurate for functions which vary gradually. The accuracy degrades for solutions which have regions of rapid variation. We introduce a dynamically varying coordinate system in which the functions vary more gradually. The optimal coordinate system is chosen from a family of explicitly varying coordinate systems by minimizing certain functionals of the solution. In our research we are testing a new family of mappings which appear to be more effective in resolving rapidly varying functions. In addition we are investigating the extension of this procedure to adaptively determine the location of multidomains. Applications will include combustion, wave propagation and fluid dynamics.

Edward A. Bogucz, Syracuse University

Two projects regarding the calculation of unsteady leading-edge flows were pursued during a four-week stay at ICOMP. The work was motivated by current Lewis interest and experiments regarding rotor-stator flows, and was conducted with close interaction with Robert J. Simoneau, James E. O'Brien, and other Lewis personnel.

The first project addressed the effects of freestream vortices on the flow and heat transfer in a stagnation-point boundary layer; this effort was a continuation of work initiated during an eight-week appointment at ICOMP in 1987. During the 1988 appointment, work was completed concerning the model problem of a circular cylinder exposed to an approaching pair of counter-rotating line vortices imbedded in an otherwise uniform crossflow. Numerical solutions for a range of conditions were obtained; these indicated that the significant parameter for this flow is the convection speed of the vortex pair at large distances from the cylinder. Vortex pairs with convection speeds characteristic of the vortices in Karman streets (which typically convect at 75% of the local freestream velocity) produce significant changes in stagnation-point flow and heat transfer, including regions of reversed flow in the boundary layer at the stagnation point and perturbations in surface heat transfer of 40% or more. The results indicate that the effects of freestream vortices may be important in understanding unsteady rotor-stator flows. A paper reporting the initial results of this work was presented at the 1st National Fluid Dynamics Congress in July 1988; a NASA Technical Memorandum describing the complete work on this problem is in preparation.

The second project concerned the numerical methods used for computing unsteady leading-edge flows. The intent of the effort is to validate methods for computing flow and heat transfer in unsteady surface boundary layers for situations involving a moving stagnation point. Work in this area was initiated during the 1988 appointment. A review of existing procedures was completed, and an exploratory project was begun concerning methods for computing boundary-layer problems on nonuniform solution meshes.

Lola Boyce, University of Texas at San Antonio

Probabilistic methods are particularly useful in the design and analysis of critical systems and components that operate in severe and uncertain environments. These methods have recently found application in space propulsion systems to improve the reliability of engine components. In order to address the reliability of such components material strength degradation as well as loading and structural analysis must be addressed in a probabilistic manner.

Material strength is affected by a number of factors, or primitive variables. These primitive variables often originate from the environment and may include stress from loading, temperature, chemical or radiation attack. Time may also interact with these primitive variables, producing effects such as creep and fatigue. In most cases strength is reduced as a result of these factors.

A constitutive equation is postulated to account for the degradation of strength due to these primitive variables. The general form of the equation is

$$\frac{S}{S_O} = \prod_{i=1}^{n} \left[\frac{A_F - A}{A_F - A_O}\right]^{a_i}$$

where A, A_F and A_O are the current, ultimate and reference values of the primitive variable, a_i is the value of an empirical constant for the ith variable, n is the number of product terms with primitive variables in the model, and S and S_O are the current and reference values of material strength. Each term has the property that if the current value equals the ultimate value, the current strength will be zero. Also, if the current value equals the reference value, the term equals one and strength is not affected. This deterministic constitutive model is calibrated by an appropriately curve-fitted least squares multiple regression of experimental data. It may then be used to predict the strength of an aerospace propulsion system component under the influence of a number of diverse effects.

The probabilistic treatment of this constitutive equation includes randomizing the deterministic equation, performing probabilistic analysis by simulation and generating probability density estimates for strength using a non-parametric method, maximum penalized likelihood. Integration yields the cumulative distribution from which probability statements regarding strength may be made. This probabilistic constitutive model predicts the random strength of an aerospace propulsion component, due to a number of diverse random effects. The information available upon which to estimate the strength, and hence the reliability, of a component is thereby increased over the deterministic estimate.

In order to calibrate the constitutive model, the values of the empirical constants, a_i, may be found by performing a multiple linear regression on the logarithm of the product terms containing the experimental data, perhaps supplemented with expert opinion. This multiple linear regression for the empirical constants and the probabilistic constitutive model are embodied in two FORTRAN programs, RAMS and PROMISS, respectively. PROMISS results are presented in the form of probability density functions and cumulative distribution functions of normalized strength, S/S_0. An example problem that considers data approximately typical of a nickel base superalloy is presented. An example of the output of PROMISS is given in the figures as density and distribution functions. RAMS results include values for the empirical constants, a_i, and associated regression statistics and diagnostics to help determine the suitability of the primitive variables and the form of the constitutive model.

REFERENCE

Boyce, L., and C. Chamis, "Probabilistic Constitutive Relationships for Cyclic Material Strength Models," Proceedings, 29th Structures, Structural Dynamics and Materials Conference, Williamsburg, VA, April, 1988.

Shih-Hung (Fred) Chang, Cleveland State University

Investigation on extending the uniformly accurate essentially non-oscillatory (ENO or UNO) shock-capturing scheme to treat the 2D Euler equations has been the main objective. The first approach was to use a Strang-type of fractional-step time-splitting method (G. Strang, SIAM J. Numerical Analysis v.5, 1968, pp. 506-517) to extend the basic 1D ENO algorithm. The resulting scheme was applied to compute numerical solutions for the shock wave reflection problem with Mach numbers ranging from 2.9 to 10.0. This approach and its implementation have been thoroughly studied. The numerical results, together with some of the 1D results, were included in a joint paper with M.-S. Liou, entitled "A Comparison of ENO and TVD Schemes." This paper was presented at the First National Fluid Dynamics Congress, Cincinnati, Ohio, on July 27, 1988.

The above study was continued and the joint paper was then expanded to an article to appear as a NASA Technical Memorandum. By the experience of this fractional-step implementation, some modified versions of the ENO scheme were proposed and studied. It was intended to improve the treatment of the cases involving high Mach numbers (greater than or equal to 5, for example). One algorithm modifies the basic ENO reconstruction procedure. The code is being developed and computations will be done shortly.

S. W. Choi, Ph.D. from Stanford University

The effects of critical layer nonlinearity on spatially growing oblique instability waves on nominally two-dimensional shear layers between parallel streams were considered in cooperation with Dr. M.E. Goldstein of NASA Lewis. The analysis shows that three-dimensional effects cause nonlinearity to occur at much smaller amplitudes than it does in two- dimensional flows. The nonlinear instability wave amplitude is determined by an integro-differential equation with cubic type nonlinearity. The numerical solutions to this equation are worked out and we show that they always end in a singularity at a finite downstream distance.

Abhisak Chulya, Ph.D. from Cleveland State University

The work under ICOMP at the Structural Mechanics Branch, NASA Lewis Research Center, is separated into two parts. Part one is the development of a new, robust finite strip plate bending element based on Mindlin/Reissner Plate theory. Part two is the development of a new integration scheme, called "Uniformly Asymptotic Implicit Scheme" for integrating the stiff differential equations of viscoplastic models for finite element analysis. Details of this work are as follows:

Part One. A linear finite strip plate element based on Mindlin/Reissner plate theory is developed. The analysis is suitable for both thin and thick plates. In the formulation new transverse shear strains are introduced and assumed constant in each two-node linear strip. The element stiffness matrix is explicitly formulated for efficient computation and computer implementation. Numerical results showing the efficiency and predictive capability of the element for the analysis of plates are presented for different support and loading conditions and a wide range of thickness. No sign of shear locking phenomenon was observed with the newly developed element.

This work has been published in 1) Proceedings of Lewis Structures Technology, May 24-25, 1988. NASA CP-3003, Vol. 2, and 2) NASA Technical Memorandum 100928, ICOMP-88-12, July 1988. It has also been sent to Computers & Structures, An International Journal for possible publication.

Part Two. To predict the response of a material using unified viscoplastic constitutive models, a system of "stiff" differential equations must be solved by numerical integration. The explicitly Euler integration scheme in conjunction with a self-adaptive strategy has been found to be efficient but it requires very small time steps. The implicit Euler integration scheme with Newton-Raphson's iteration is more stable and accurate for both very small and very large time steps, but consumes more CPU time and is less accurate in the region where the curvature of the stress-strain relation is large.

The objective of this work is to present a new asymptotic implicit integration scheme and demonstrate the potential of the scheme for the analysis of high temperature viscoplastic constitutive models. Most unified viscoplastic differential equations can be transformed into integral forms which are then expanded in a uniformly valid asymptotic series. The stress and state variables over the time step increment are obtained by using the implicit recursion scheme coupled with Newton-Raphson iteration.

The algorithm is incorporated into a MARC finite element program by subroutine HYPELA with three unified viscoplastic constitutive models, namely Walker's , KSR's, and Miller's. Numerical examples are tested on uniaxial hysteresis loop with isothermal condition, uniaxial case with nonisothermal condition, and combustor liner analysis. Comparisons are also made with the self-adaptive explicit Euler integration scheme. Results show the superiorities of the new integration scheme both in accuracy for a larger time step and in computational efficiency. Finally, the algorithm is applied to the continuum damage mechanics model of Walker. Result shows the tremendous advantage of this scheme over the explicit Euler integration scheme, especially near damage.

A writeup of this work is in process for possible publication in: 1) NASA Technical Memorandum under ICOMP, 2) International Journal for Numerical Methods in Engineering, and 3) Journal of Applied Mechanics.

S. J. Cowley, Imperial College of Science and Technology

Four problems were worked on during the visit; they are

(1) Work on singularity formation in solutions of the unsteady triple-deck equations was largely written up. Using a pseudo-spectral method, together with a filter to control rounding error, accurate numerical calculations have been performed to demonstrate that the triple-deck description of Tollmien-Schlichting waves only has a finite time of validity, i.e. a singularity develops in the Tollmien-Schlichting wave solution after a finite time.

(2) Work on singularity formation in the classical unsteady boundary-layer equations was written up. This is joint work with L. L. van Dommelen, and extends his earlier work from two to three dimensions. In addition a certain amount of effort was spent writing a code to solve the three-dimensional unsteady boundary-layer equations.

(3) Work with P. Hall on the stability of hypersonic flow past a wedge was written up. The interest here is in the interaction between Tollmien-Schlichting (or Rayleigh) waves and the shock. A talk was given on this work.

(4) Calculations were performed on a model boundary-layer problem connected with large-Reynolds-number flow past a circular cylinder. The aim was to study marginal separation over a moving wall, and to see whether a recently reported theory by Sychev was correct (verdict still out).

W. N. Dawes, Cambridge University

ICOMP activity has been mainly in collaboration with Jerry Wood in the ongoing evaluation of my 3D turbomachinery Navier-Stokes code (soon to be acquired by NASA). This activity has taken the form of a series of studies concerned with the implementation of the turbulence model and the artificial viscosity formulation.

The code uses the well-known two layer model proposed by Baldwin and Lomax. Their published model constants were tuned to give good agreement with zero pressure gradient data. Recently work has been published showing how the Baldwin-Lomax constants might be modified to account for pressure gradient effects. Accordingly, I carried out a thorough review of the relevant boundary layer literature and arrived at an intriguing conclusion. The Baldwin-Lomax constant CCP should indeed reduce with increasingly adverse pressure gradient. However, the Clauser constant CCL is in fact not constant but increases with increasingly adverse pressure gradient. The net result is that the combined constant (which is all that counts in the Baldwin-Lomax formulation) given by CCL*CCP is in fact constant (to $\pm$5%) at 0.0168*1.2 over all pressure gradients up to separation.

Secondly, the artificial viscosity formulation used by the 3D code was studied. Artificial viscosity is used in a code to control odd-even point solution decoupling and to aid shock capture. My 3D code uses the well-known combination of fourth and second differences, blended via a pressure gradient switch. My implementation in the code presumes that near no-slip surfaces (blades, hub and shroud) the physical viscous forces are large enough for artificial viscosity to be not necessary; accordingly none is coded. Nevertheless, the use of artificial viscosity in the core flow can give rise to spurious entropy production. By suitable scaling of the magnitude of the artificial viscosity terms to take into account large variations in cell aspect ratio the general level of spurious entropy production has been reduced (from an already rather low level).

Ammar Degani, Ph.D. candidate, Lehigh University

In the conventional computation of turbulent boundary layers, a substantial number of grid points are required to adequately resolve the intense variation of velocity and temperature with the inner layer. In wall-function methods, however, the velocity and temperature within the inner layer are obtained from analytic expressions (e.g., "the law of the wall"), and the solution in the outer layer is obtained numerically by adopting a suitable closure model and is matched to the inner layer solution at the first grid point from the wall. Typically, about 50% of the total number of grid points that would normally be used in a conventional calculation can be eliminated by using the wall-function method without any degradation in accuracy. This method, however, has so far been restricted to two-dimensional flows, but my current research focuses on developing a method to incorporate the wall-function method in the computation of three-dimensional fully turbulent flows.

At ICOMP this past summer, I worked on obtaining the asymptotic (in the limit of large Reynolds number) structure of a three-dimensional turbulent boundary layer with heat transfer and obtaining algebraic relationships, which determine the shear stress and heat transfer at the wall, from the matching of the inner and outer solutions. I also worked on devising and implementing a numerical procedure for the wall-function method.

A. O. Demuren, University of Lagos

Turbulent jets in crossflow are found in many engineering applications such as internal and film cooling of turbine blades, fuel/air mixing in gas-turbine engines, vertical and short take-off and landing (V/STOL) aircraft in transition flight, etc. In most cases, the flow is turbulent and three-dimensional. Accurate numerical simulation with existing methods has not been possible because of inadequate spatial resolution and excessive computational time requirements on the one hand and deficiencies in turbulence modelling on the other.

My research consists of two parts:

(1) To overcome the problem of spatial resolution and excessive computational time, through a combination of multigrid acceleration and adaptive grid techniques. The characteristics of various multigrid procedures are now being analyzed in combination with successive over-relaxation and conjugate-gradient methods. Advantages and disadvantages of using non-structured grids are also being investigated.

(2) A focus on the turbulence model. Most simulations have used the eddy viscosity concept (coupled with the solution of differential equations for k and ϵ) to model the turbulent stresses in this flow, but recent measurements (e.g. Andreopoulos and Rodi, JFM 1984) show that this may be inaccurate in many parts of the flow, especially in the wake region. The procedure for implementing the solution of the full Reynolds stress equations in the 3D jet calculations is being developed. In order to combine the multi-grid procedure effectively with the turbulence model, the usual practice of representing turbulence boundary conditions with wall functions must be discarded in favor of integration through the sublayer right down to the wall.

Peter W. Duck, University of Manchester

My first activity (jointly with Dr. Goldstein) was in initiating an investigation of the effect of a plane, traveling wave encountering a boundary layer on a semi-infinite flat plate in a supersonic flow. The initial aim was to investigate the so-called "receptivity" problem, i.e. given some disturbance in the freestream flow, of specified amplitude, (such as an acoustic wave) to find the amplitude of any associated instability waves in the boundary layer. The problem evolves downstream from an unsteady boundary layer type of flow (near the leading edge) to an inviscid (Rayleigh) flow involving transverse pressure gradients further downstream. However, it transpires that the problem for the inhomogeneous component of the solution, far downstream, has not been properly analyzed, and much of the research effort was focussed in this direction.

A computer code for solving the Rayleigh problem (including the effects of forcing) has been developed and has aided the analytical treatment of this problem.

My second activity has been concerned with three-dimensional marginal separation. Here the problem is to consider a steady, laminar, incompressible boundary layer which almost/just separates at a point. The two-dimensional problem has been considered by S. N. Brown, F. T. Smith, K. Stewartson, and others, and is now reasonably well understood.

In the three-dimensional case, the analysis leads to a nonlinear integral equation (in two space variables). Although the previous (two-dimensional) work implemented numerical schemes in physical space to solve the appropriate integral equation (in one space variable), my approach has been to use a pseudo-spectral approach to solve the three-dimensional case. Here the solution is achieved partly in double Fourier space and partly in physical space, with an F.F.T. routine being used to transform the solution from one space to the other. This type of technique appears to have a number of significant advantages over previous schemes based entirely in physical space. An interesting and important phenomenon found in the present three-dimensional computations (and also the previous two-dimensional work) is that nonuniqueness of solution is possible. A paper describing this work is currently in preparation.

George S. Dulikravich, Pennsylvania State University

I performed research on physically based artificial dissipation modeling during my stay at ICOMP/NASA Lewis last May. I worked on the actual implementation of a new Physically Based Dissipation (PBD) model in my two-dimensional steady transonic full potential analysis code. The transonic shocked flow results are comparable to those obtained with the conventional artificial density and artificial viscosity formulations without any oscillations in the vicinity of the shock.

A second topic that I formulated analytically involved higher order dissipation based on the physical dissipation due to radiation heat transfer. Corresponding boundary conditions were also formulated that are to match the higher order derivatives in the dissipation model. Discussions with Dr. Robert Siegel of NASA Lewis Research Center concerning the basic assumptions used in the existing models of radiative heat transfer were very useful in this work. A third topic, that I discussed with Dr. van Leer of the University

of Michigan who was visiting ICOMP, involved a new Distributed Minimal Residual (DMR) algorithm developed by my research group that is capable of significantly accelerating iterative convergence of explicit schemes for stiff systems of partial differential equations.

Peter R. Eiseman, Columbia University

My activities have included a lecture series on algebraic grid generation, the development of an interactive consultation with both ICOMP and NASA personnel, and participation in the NASA Lewis internal review process for papers. To be more specific, the lecture series occurred as an earlier activity and is likely to be continued for the topics of adaptivity and surface grids in 3D. Those new topics arose in the course of various technical discussions with ICOMP and NASA personnel. Such discussions have also included various strategies for numerical fluid dynamic simulations as well as for the physical problems of interest to NASA Lewis researchers. While numerous discussions have also taken place within the topic of grid generation, there has also been a direct and active involvement in the development of the interactive grid code for turbomachinery. This has involved the guiding of work by others as well as performing many specific technical tasks. The principal NASA co-worker on this development is Yung K. Choo. Also included are Charles Reno and some new additions to be brought in by Dr. Choo.

A major part of the activity for the turbomachinery grid code has now been assumed by the NASA Lewis Grant NAG3-877. While that grant has supplied a much needed support, the additional ICOMP support has been particularly important as well. Through ICOMP, the communications created by an enhanced personal presence has been indispensable. It has not only helped in the technical interactions with Dr. Choo and collaborating colleagues, but also in establishing a general awareness among other NASA Lewis investigators that there are potential benefits to be gained from the grid code. Without such an awareness of the current and future capabilities being given on a continuing basis, the turbomachinery grid code would probably not receive wide spread use at NASA Lewis, regardless of how good it might be.

Graham K. Ellis, M.S. from Virginia Polytechnic Institute and State University

An extensive series of performance analyses were undertaken on the NASA Lewis transputer parallel processing system in order to determine how to program the system for maximum network efficiency. The results of these studies indicate that problems must be of sufficient size to justify implementing them on a parallel network. When a network is programmed properly, efficiencies of 99.8 per cent have been measured. This corresponds to a speedup of 39.9 for a 40 processor network.

A general purpose two-dimensional graphics package was developed for the transputer based parallel processing system. These general purpose routines are designed to be easily incorporated into any programs and full documentation is available.

The transputers in the parallel processing laboratory have been upgraded with floating-point processors. These new processors provide approximately an order of magnitude performance increase over the older chips.

Mohammad Farshchi, Nielsen Engineering & Research, Inc.

I spent six weeks as a visiting researcher at ICOMP. My present research interest is the development of turbulence and combustion models for supersonic reacting flow fields. I have utilized the probability density function (PDF) approach for gaining insight and modeling of pressure-velocity interactions in compressible flow fields. The PDF approach is also ideal for modeling of chemical reacting source terms in species conservation equations and I am working on extending this methodology to compressible reacting flows. A major portion of my time at ICOMP was devoted to adding a turbulence model to computer code (RPLUS). The code is a Navier-Stokes solver with finite rate chemistry and uses a lower-upper symmetric successive overrelaxation scheme. The turbulence model being coded is a second-order Reynolds stress model which accounts for the effects of compressibility through inclusion of mean dilatation terms. The model accommodates low Reynolds number regions and equations are integrated all the way to the wall. I also familiarized myself with the CRAY2 computer system at NAS facilities, which will be used to run the above code.

Michael R. Foster, Ohio State University

In some previous work, F. T. Smith and I[1] have investigated the inertial helical instabilities of Long's similarity solution for a slender vortex with axial flow[2] in a limit in which the axial momentum flux is much larger than the axial flux of angular momentum. (The large parameter for this asymptotic state is the dimensionless "flow force", denoted by M). We found that this vortex in one such case (the "Type II" vortex) consists of a sech^2 axial jet profile wrapped in a ring around a weakly counterflowing central core. Across this jet, the swirl grows rapidly from nothing to that for a potential vortex. At axial wavelengths of the order of the vortex width, the instability modes are of two kinds, namely, the Bickley-jet sinuous varicose modes. Modes with long axial wavelengths, however, differ substantially, owing to the increased importance of azimuthal convection of such disturbances. Hence, any neutral-stability criterion is modified.

Over the summer at NASA Lewis and into the fall here at Ohio State, I have been placing this inviscid instability theory in the context of high Reynolds number asymptotics, to endeavor to understand, for these long-wave modes, any stabilizing effects of viscosity. The result is that viscosity does indeed stabilize the sinuous mode. We find no effect for Reynolds numbers larger than $M^{5/2}$; however, for Reynolds numbers of that order, the eigenfunction structure is dramatically altered, with the growth rate vanishing at a particular value of $Re/M^{5/2}$, for a mode with axial wave length of order M^3. The particular value of the critical Reynolds number and wave length await numerical calculation.

In a related effort, Smith and I are formulating the asymptotic instability character for the large-M, "Type I" Long's vortex; over the summer, a series of computations was made at NASA Lewis to determine numerically the nature of the instabilities in this case, to begin to compare with the analysis underway simultaneously. There were some surprises in the numbers obtained requiring a more careful look at the analysis, giving yet another example of the synergy between numerical and analytical approaches to a particular problem.

REFERENCES

[1]Foster, M. R. and Smith, F. T.: "Stability of Long's Vortex at Large Flow Force," submitted to Journal of Fluid Mechanics in revised form, September, 1988.
[2]Long, R. R.: "A Vortex in an Infinite Fluid," Journal of Fluid Mechanics Vol 11, pg. 611, 1961.

Jitesh Gajjar, Exeter University

Cross flow instability is characteristic of many three-dimensional boundary layer flows, for example in flow past swept aircraft wings, rotating disk flow, and rotating axisymmetric bodies. Hitherto much of the effort has concentrated on studying the stationary cross flow vortex but recent experiments by Arnal et al (1987), Bippes (1986) indicate the importance of the non-stationary waves in the transition process.

I looked at the stability of both the stationary and non-stationary waves. The problem for the stationary cross flow vortex is much easier and it was shown that the evolution of the stationary modes is governed by a nonlinear amplitude equation which is coupled with the solution of an unsteady nonlinear critical layer together with a linear wall layer problem. A similar investigation for the nonstationary waves is still in progress.

Another problem considered was the solution of the nonlinear composite critical layer equations arising in the stability of two dimensional compressible boundary layer flows. The neutral eigenrelations depend crucially on the properties of the nonlinear critical layer, and the phase shift across the critical layer turns out to be significantly different for the compressible case than the usual Haberman problem. A paper on this work is in preparation.

Peyman Givi, State University of New York at Buffalo

The expected development of a family of airbreathing hypervelocity vehicles during the next few decades has motivated a dramatic increase in research activities in the area of supersonic reactive flows (Northham, 1984). Of particular interest are the reactive systems of Supersonic Combustion RAMJET engines (SCRAMJET) and the Hypersonic Dual-Combustion Ramjet (HDCR) [Waltrup, 1986; White et al., 1986]. A survey of the airbreathing technologies for hypersonic application can be found in a recent article by Waltrup (1986). According to this review, in the combustor of a scramjet hydrogen fuel is injected at near sonic conditions into a supersonic air stream and combustion occurs in the recirculatory mixing zone inside the combustor. The analysis of such high temperature flow is complicated by the interaction between fluid mechanical and chemical effects, where the reactants first mix and then combine to release heat which, in turn, alters the flow field significantly. A basic understanding of such complex physical phenomena has created a renewed interest in the area of high-velocity reactive flows.

Among the computational work on supersonic combustion, a recent dissertation by Drummond (1987) is to be mentioned. In this work, a hybrid pseudospectral-finite difference is employed to simulate a spatially developing reacting supersonic mixing layer. Calculations are also performed by employing a finite-difference algorithm for the purposes of numerical comparisons. The mixing region formed at the trailing edge of a splitter-plate configuration is the subject of numerical investigation. This

configuration involves a hydrogen-air mixture (similar to that to be supposedly used in scramjets) with fuel and oxidant initially separated by a finite thickness splitter plate. The Mach numbers in both streams are 1.5; therefore the shocks (if any) formed within the layer are very weak. In the finite-difference simulations, the fuel and the oxidizer are separated by a finite thickness splitter plate. In the hybrid pseudospectral-finite difference simulations, however, the plate is not included, but its influences are retained by imposing a smoothed streamwise velocity profile at the inlet of the layer. Hydrogen is introduced into the mixing zone at room temperature, while oxygen is heated to 2000 °K to initiate chemical reactions. Calculations are performed for both reacting and non-reacting layers in order to assess the influences of chemical reactions on the supersonic layer.

The simulated results confirm that the supersonic layer is much more stable than a similar layer with subsonic streams. In a high Mach number flow, the rate of growth of the shear layer thickness is not very large and it takes longer to respond to imposed perturbations triggering its growth. This is evidenced from the vorticity contour plots of the finite-difference simulation (presented in Drummond's thesis) which shows that the vortical structures of the layer are not nearly as distinct as those of subsonic flows. The lack of structures is evidenced more clearly in the hybrid pseudospectral-finite difference simulations, as the layer is not triggered by the imposed initial velocity condition. Drummond (1987) tried to trigger the flow by superimposing a perturbation on the mean flow expressed by:

$$U(X = 0) = U_m + U' \qquad (1)$$

where U_m represents the mean velocity, and U' is selected in such a way to closely match the velocity fluctuations found at the inlet of the layer in the finite-difference simulations. The computed perturbation approximately correlated a single perturbation function of the form

$$\frac{U'}{U_m} = A \exp(-aY^2)\sin(\omega t) \qquad (2)$$

where A,a, and ω are computed by analyzing the data generated by finite-difference simulations. X and Y denote the streamwise and the cross-stream coordinates, respectively. Other aerothermodynamic variables are also triggered in a similar fashion. The vorticity contours of the layer in this case again indicate that the layer is not growing substantially and the imposed disturbances do not appear to be amplified significantly. With an exception of a close region near the inlet, the vorticity contours form parallel undisturbed lines which are similar to those of an unperturbed convectively stable parallel layer. Nevertheless, the perturbations imposed by Eq. (2) result in the enhancement of mixing and reaction at a region further downstream of the layer. This is probably due to the flapping oscillation of the layer due to sinusoidal perturbations at the inlet which assists in enlarging the reaction surface area of the two reactants.

A comparison is made between the simulated results obtained with and without the influences of chemical heat release in the finite-difference simulations. This comparison shows that the amplitudes of the disturbances in the reacting calculations are substantially lower than those of the non-reacting case. This is demonstrated by a comparison of velocity fluctuations in a different location of the layer. The comparison indicates that the presence of exothermic reaction suppresses the velocity fluctuations which, in turn, leads to a lower growth rate of the compressed layer. This behavior is very similar to those observed in the previous subsonic calculations of McMurtry (1988) and the measurements of Hermanson (1985), who showed that heat release has a substantial influence on the retardation of mixing. (See Givi, 1988 for a recent review). However, since the simulated flow is two-dimensional, a realistic coupling between small scale three-dimensional turbulence and exothermic reaction are not provided.

The work of Drummond (1987) is a first step in understanding the mechanism of mixing (or lack of mixing) in supersonic free layers. However, the calculations are substantially complicated due to inclusion of complex kinetics (9 species, 18 reactions). Such complex reactions may not allow a fundamental understanding of some of the less complex, but still unknown, problems.

The main objective of our work initiated at ICOMP is to establish a better understanding of the mechanisms of mixing and chemical reactions in supersonic reacting parallel shear flows. As an initial effort, the phenomena of mixing in a two-dimensional layer has been the subject of our numerical investigation; while in the long run, we plan to extend the analysis to investigate three-dimensional supersonic flows under the influence of finite rate chemical reactions. Due to severe computational requirements, however, the type of chemical reactions to be included, will not be more complex than one described by a single-step kinetic mechanism. In the first phase of this study, we have employed MacCormack's second-order explicit scheme to simulate a non-reacting two-dimensional layer. The results of the simulations exhibit the roll-up and pairing of the vortical structures caused by imposed perturbations at the inlet of unsteady shear layer. However, it was found that one must be very careful in utilizing the MacCormack scheme for this type of simulation in that like other shock capturing schemes, the results are dependent on the number of numerical grid points. Simulations were performed with different grid points and with different values of the Reynolds (Peclet) numbers in order to ensure the accuracy of the results on the number of adopted grid points. An attempt is initiated to improve the accuracy of the scheme by upgrading the accuracy of the spatial differencing to fourth order, which would certainly help in our future calculations. The results obtained so far by the use of the second-order method indicated that there are a large number of structures formed in the spatially developing shear layer. This complicates the subsequent analysis associated with the interactions between the vortices and the shocklets formed in the compressed layer. Therefore, the simulations of a temporally developing layer describing the transport of a single isolated vortex is undertaken to study such interactions in a less complex configuration. We are now at the final stages of debugging the computer code to be used for this calculation. Our future efforts will also involve the use of other numerical methods (e.g. TVD (Yee, 1987) to name one) for the simulations of compressible turbulent reactive flows.

REFERENCES

Drummond, J. P. (1987), Ph.D. Thesis, George Washington University, Washington, DC.
Givi, P. (1988), Prog. Energy Comb. Science, to appear.
Hermanson, J. C. (1985), Ph.D. Thesis, Cal-Tech, Pasadena, CA.
McMurtry, P. A. (1988), Ph.D. Thesis, University of Washington, Seattle, WA.
Northham, G. B. (1984), JANNAF 21st Combustion Meeting.
Waltrup, P. J. (1986), AIAA Paper AIAA-86-0158.
White et al. (1986), AIAA Paper AIAA-86-0163.
Yee, H. (1987), NASA Technical Report, NASA Ames Research Center

Murli M. Gupta, George Washington University

I have developed single cell high order schemes (SCHOS) for certain elliptic equations, especially the convection-diffusion equations. These schemes have truncation errors of high order (often fourth order), are of compact type thus requiring no special treatment near the boundaries, and are stable thus facilitating the use of iterative solvers. During my visit at ICOMP, I extended the fourth order SCHOS schemes to the Navier-Stokes equations. As a test problem I computed steady-state incompressible viscous flows in a driven cavity. In addition to approximating the streamfunction and the vorticity equations by fourth order difference schemes, I also experimented with the calculation of velocities using second and fourth order difference approximations. Numerical solutions using Cray XMP24 were obtained for Reynolds numbers ranging from 1 to 2000 for a 41x41 mesh. The numerical solutions were found to compare very well with the benchmark solutions available in the literature. In particular, I discovered that the use of fourth order approximations for velocities, as compared with the use of second order approximations, provides better accuracy at a lesser cost.

I initiated a project to apply the multigrid techniques to elliptic equations, the eventual aim being to apply to the Navier-Stokes equations in conjunction with the fourth order difference schemes. Preliminary results are quite encouraging and I expect substantial progress in the near future. I also refined my code for spectrum enveloping algorithms to accelerate the convergence of an already convergent iteration and to obtain convergent solutions from a divergent iteration.

The visit at ICOMP was very fruitful in terms of the results actually obtained as well as in the seminars attended and discussions held with other ICOMP visitors and the NASA Lewis researchers.

Thomas Hagstrom, State University of New York at Stony Brook

The major focus of my work at ICOMP was experimentation with and development of asymptotic boundary conditions for computational fluid dynamics. The asymptotic approach typically requires different techniques for different problems. We have considered two particular cases: inviscid, compressible flow in an exterior domain and viscous, incompressible flow in a channel.

For the compressible case, which is being studied in collaboration with S. I. Hariharan, we are assuming a uniform flow in the far field. This is a generalization of work carried out last year at ICOMP where the gas was assumed to be at rest and isentropic at infinity. We have analyzed the linearized Euler equations and developed appropriate asymptotic expansions.

Seconding the observations of other workers, we find that vorticity, entropy, pressure and divergence are the most convenient variables to work with. As part of our analysis, we have developed new far-field expansions of outgoing waves for anisotropic wave equations. The conditions are presently being tested on a variety of problems. Future work should include a study of nonlinear effects.

The study of incompressible channel flows is also a continuation of work carried out last year. We have developed a new formulation for stream function variables as well as for primitive variables. Working with J. Goodrich, tests of these and other conditions are being undertaken for a variety of channel flow problems. Experiments involving transitional flows have also been discussed with J. Gajjar and may be started in the near future. The next step in this effort is the extension to the compressible, Reynolds-averaged Navier-Stokes equations.

Finally, some discussions were held with A. Bayliss concerning a boundary condition problem arising in his simulations of combustion phenomena. Also, we did some work on analytical models of low frequency, finite amplitude perturbations of Poiseuille flow.

S. I. Hariharan, University of Akron

Calculations of unsteady vortical disturbances around a flat plate which is modeled after an airfoil are under investigation. The problem that results in this process is governed by a hyperbolic equation in an open domain. For computational purposes the open domain must be truncated by finite artificial boundaries. The open domain problem, when truncated into a finite domain by these artificial boundaries, requires approximate boundary conditions that simulate the behavior at infinity. Approximate boundary conditions have been derived. Well-posedness of the problems in the truncated region are also shown. A numerical solution procedure for the time domain calculation has been derived. Comparisons are made, with solutions of the frequency domain problem done by Scott and Atassi, and good agreement is shown. The stability of our numerical scheme is discussed, and the stability restriction for the scheme is established as a function of the Mach number. This work is being carried out in conjunction with Mr. Yu Ping (graduate student, The University of Akron) and with J. R. Scott of Lewis Research Center.

Other ongoing work at ICOMP is on boundary conditions for compressible flows done in conjunction with Dr. Thomas Hagstrom of State University of New York at Stony Brook. Please see details in his progress report.

Giusseppe S. Iannelli, Ph.D. candidate, University of Tennessee

During my one-month stay at ICOMP I have examined a finite-element-method for the Euler and Navier-Stokes equations. This procedure incorporates a Taylor weak statement dissipation mechanism and a versatile Runge-Kutta implicit time integration algorithm framework. Two dissipation mechanisms were considered and implemented and the more efficient one was identified.

The retention of the static pressure in the Euler equations provided a simple yet accurate way to enforce the exit pressure boundary condition. Of comparable importance a suitable combination of the weak statement flux-vector surface-integrals has allowed a convenient non-discrete way to enforce the fundamental wall-tangency boundary condition, in the Euler equation. A

similar association of the viscous-term vector surface integrals has permitted to enforce, again in a non-discrete way, Dutt's maximally dissipative boundary conditions, in the complete Navier-Stokes equations.

The detailed derivation of two implicit non-linearly stiffly stable and optimally 2nd order accurate Runge-Kutta algorithms has been presented. These alternative time-integration procedures have been theoretically compared with the conventional backwards Euler and Trapezoidal rules. The chief findings were that the possibility exists that convergence or a steady state may not be achieved with the latter integration procedures, for the non-linear model problem considered, whereas the examined Runge-Kutta methods always converge to a steady state.

The accuracy, dissipation-mechanism performance and shock-capturing capability of the FEM procedure investigated have been preliminarily assessed by numerically solving an informative quasi-one-dimensional inviscid-flow problem in a de-Laval nozzle at off design conditions. The results achieved were monotone and compared quite well with those generated by Liou who utilized a flux vector splitting approach to solve the same problem.

The procedure examined appears theoretically robust and accurate. The utilization of uniform centered spatial discretization in concert with the dissipation mechanism derived seems to exercise adequate control over discretization dispersive and out-diffusive errors. In a future phase this FEM procedure will have to be tested for the Navier-Stokes equations. Moreover a theoretical framework will have to be developed that could supply a guide in the non-empirical choice of the dissipation-mechanism parameter.

Bo-Nan Jiang, Ph.D. from University of Texas, Austin

It has been more than two decades since people started applying finite element methods to fluid dynamics problems. Subsequently, a large number of papers and many books on finite elements in fluids have been published, and numerous fluid dynamics problems have been successfully solved by finite element methods. Currently, however, only two commercial general-purpose finite element packages for incompressible flows, i.e. FIDAP and FLOTRAN, are available. This situation differs from that in solid mechanics. Twenty years after the first finite element papers were published, several dozen finite element commercial packages for solid mechanics problems were on the market. Perhaps, one reason for the lack of fluid dynamics finite element codes was that there was no unified method which could cover a wide range of fluid problems. For example, the classic Galerkin method is used for potential flows, the mixed Galerkin method and the penalty method are dominant for incompressible viscous flows, the Taylor-Galerkin method and the Petrov-Galerkin method are developed for convective transport problems and compressible flow problems. Because the principles and structures of these methods are different, it is extremely difficult to implement these methods in a general-purpose code.

The objective of my research (in cooperation with Dr. L. A. Povinelli) is to develop a unified method, namely the Least-Squares Finite Element Method (LSFEM), for fluid dynamics problems. For incompressible Navier-Stokes equations, LSFEM leads to a minimization problem rather than a saddle problem. Hence, LSFEM can circumvent the Ladyzhenskaya-Babuska-Brezzi test and accommodate equal-order interpolations, which can be conveniently

implemented in a general-purpose program. For the compressible Euler equations, LSFEM naturally generates numerical dissipation without any added parameters to capture shocks.

This work will be continued to extend LSFEM to solving three-dimensional problems by using the element-by-element preconditioned conjugate gradient method which is very suitable for vector and parallel supercomputers.

Sang-Wook Kim, Ph.D. from University of Texas, Arlington

A near-wall turbulence model and its incorporation into a multiple-time-scale turbulence model have been studied. In the method, the conservation of mass, momentum, and the turbulent kinetic energy equations are integrated up to the wall; the energy transfer rate and the dissipation rate inside the near-wall layer have been obtained from algebraic equations. The algebraic equations for the energy transfer rate and the dissipation rate inside the near-wall layer have been obtained from a k-equation turbulence model and the near-wall analysis. Fully developed turbulent channel flow and fully developed turbulent pipe flow were solved using a finite element method to test the predictive capability of the turbulence model. The computational results compared favorably with experimental data. It was also found that the turbulence model could resolve the over-shoot phenomena of turbulent kinetic energy and the dissipation rate in the region very close to the wall.

In a second study, control-volume based finite difference code to solve the Navier-Stokes equations at all flow velocities (i.e., from Mach number zero up to hypersonic flows) has been developed. The code has been tested for (1) developing channel flows, (2) developing pipe flows, (3) a 90-degree bent two-dimensional channel flow, (4) a turbulent flow over a curved hill, and (5) a turbulent boundary layer-shock wave interaction flow in a supersonic compression corner. Computation of a shock-wave induced flow separation for transonic flows over a curved hill is in progress. A partially low Reynolds number $k-\varepsilon$ turbulence model has been used in these computations. For the example flows considered, the computational results compared favorably with available numerical and/or experimental data.

W. Kollmann, University of California, Davis

The research work carried out during the visit at NASA Lewis was focussed on turbulence in compressible supersonic flows. Previous attempts at analysis of this type of flow lead to the question of the statistical description of discontinuities embedded in smooth flow fields. A single scalar variable was considered and the statistics of the values of this scalar were described in terms of the probability density function (pdf) at a given point in the flow field. The dynamical equation for the scalar at a fixed point in space was set up in general fashion as a stochastic differential equation containing differentiable, continuous and discontinuous contributions. The resulting transport equation for the pdf shows, that the discontinuous part of the scalar dynamics appears as an integral term in the pdf equation and the pdf varies smoothly in contrast to the scalar.

Lala B. Krishna, University of Akron

My research, conducted with Joseph Padovan of the University of Akron, is to develop new parallel computer solution algorithms of large scale simulations drawn from the interdisciplinary areas of computational mechanics. The initial phase of the work involves the linear and nonlinear simulations of the problems of the size 10^5-10^9 equations. This will involve developing direct, interactive, and mixed direct iterative type solvers. A hierarchical partitioning procedure will be employed to parallelize the solution.

Our model problems include elliptic solvers and solution of heat conduction problems. To improve the stability and robustness of the scheme, the classic approach of approximating differential operators via difference schemes will be replaced by rational approximations. Currently we are deriving improved Chebyshev rational functions to approximate the differential operators using a multiple precision technique on a Sun Micro computer

B. P. Leonard, University of Akron

During the summer 1988 contract period, the initial phase of the ULTIMATE project (universal limiter for transient interpolation modelling of the advective transport equations) was completed and appears as NASA TM-100916. Extension of the scheme to nonlinear one-dimensional problems has been set up and applied to a Burgers-equation test problem. As expected, this shows very-high-accuracy nonoscillatory results, capturing the shock in two mesh-widths (the minimum possible). As a further prototype of systems of nonlinear equations (such as the Euler or Navier-Stokes equations), a simple hydraulic-jump test problem has been formulated. Initial results confirm my belief that gasdynamic (shock-capturing) simulation can be achieved without resorting to the many complexities associated with accepted (TVD-type) methods (such as flux-vector or flux-difference splitting, explicit calculation of eigenvalues, approximate Riemann solvers, etc.), based on implicit (or explicit) artificial-viscosity techniques. This, of course, has particularly important ramifications for practical 3-D simulation. The key to this major advance lies in using higher order (artificial-viscosity-free) schemes constrained by the universal limiter.

Papers concerning this work, and previous ICOMP work on the ULTRA-SHARP project (universal limited for tight resolution and accuracy using a simple high-accuracy resolution program) were presented as follows:

(i) "Sharp Numerical Simulation of Thin Layers in Highly Convective Flows" at the BAIL-V International Conference in Shanghai in June 1988.

(ii) "Universal Limited for High Order Explicit Conservative Advection Schemes" (with H. S. Niknafs) at the 11th International Conference on Numerical Methods in Fluid Dynamics in Williamsburg in July 1988.

(iii) "Third-Order Multi-Dimensional Euler/Navier-Stokes Solver" at the First National Fluid Dynamics Congress in Cincinnati in July 1988.

These appear in their respective proceedings, giving appropriate ICOMP affiliation. I also presented a paper, simply called "ULTIMATE" at the APS-DFD meeting in Buffalo in November 1988. Finally, I have prepared a short paper "Note on the Formal Accuracy of TVD Schemes" which I am submitting to the Journal of Computational Physics. This is likely to be quite controversial, as it demonstrates that TVD schemes are second-order accurate at extrema--not first order, as commonly believed.

Avi Lin, Temple University

Since this is my first visit to ICOMP my activity during this stay was mainly influenced by the various activities projects and programs in the Internal Fluid Dynamics Division.

My first activity was in the appropriate mathematical consistency of the various discretized versions of the Navier-Stokes (NS) equations. I have proved that for the steady-state incompressible NS equations appropriate boundary conditions should be supplied only for three variables (which are usually the velocities) while for the other variable (normally the pressure) a value should be specified only at a point in the field. This result leads to a double staggering discretization of the variables (means staggering the variables as well as the equations over the discretization stencil), proving that the discretized operator is singular when all the variables are defined at the same grid point. Using this result I have solved "mathematically correctly" the Poisson equation using the Cauchy Riemann primitive variables, getting residues which are better than the convergence criteria. I have also solved the two dimensional NS equations in a simple domain using a similar strategy, with the Blocked Strongly Implicit procedure, and have gotten similar results.

My second activity was in the parallel CFD area. Here I have proven the existence and uniqueness of iterative parallel algorithms for the incompressible NS equations. The idea is based on a new operator we have developed, namely, a "parallel splitter" operator (as opposed to a flux splitting operator). All of these are only theoretical results that have to be implemented on parallel machines for verification. Part of these results were presented to the IFIP Working Group 2 meeting at Stanford University in August 1988.

My last activity was in turbulence modeling. I have adopted the Turbulent Tensorial Volume model that was formulated by me in 1980. I have shown that this model does not require any special "wall functions" that other models (like the k-ε) need, yet brings a lot more physics and understanding of the problem. For example, it will never produce negative values of k or ε (as with some of the k-ε models), yet it does contain a small number of coefficients compared to other models.

I have enjoyed very much this first visit to ICOMP. The seminars and the workshops were excellent. My discussions with the other ICOMP visitors as well as the Lewis people were fruitful. I hope to continue these above initiatives in my forthcoming visits.

Jong-Shang Liu, Ph.D. from Case Western Reserve University

The complicated flow phenomena due to rotor/stator interaction in turbomachinery have been extensively investigated. A Runge-Kutta scheme with residual smoothing is adopted to solve the unsteady Euler equations. Good computational efficiency is achieved by reducing the calculated domain to only one flow passage. The problem of uneven rotor/stator pitches is resolved by using a tilted time domain technique. Numerical results are obtained for flows over a NACA 0012 cascade and the first stage of the fuel turbopump of the Space Shuttle Main Engine. The present study will be continued to validate the numerical results.

Reda R. Mankbadi, Cairo University

Excited jets have been the subject of several recent investigations. These investigations are concerned with the fundamental understanding of the jet development as well as possible technological applications such as control of the jet mixing. In order to make flow control techniques technologically attractive it has to be demonstrated that this "control" can be exerted in "real life flow," which unlike the laboratory flows has higher levels of free stream turbulence. It is also of fundamental interest to understand how the initial turbulence controls the growth of stability waves in excited shear layers. The present work is therefore concerned with examining the effect of the initial turbulence on the development of excited jets.

In the present work, Mankbadi and Liu's analysis (Phil. Trans. Roy. Soc. of London, Vol. 298, pp. 541-602, 1981) is extended to account for a variable core turbulence. Phase-averaging along with the conventional time-averaging techniques are applied to the full incompressible momentum equations to obtain the governing momentum equations for the mean flow, the imposed stability component and the turbulence component. These equations are used to obtain the integral kinetic energy equation for each flow component. The profile of the stability component is obtained by solving the boundary value problem of the locally parallel stability equations. The profile of the turbulent stresses are taken to be Gaussian with a peak at the center of the shear layer. Within the potential core the turbulence level is equal to the initial core turbulence. The resulting set of equations are solved for several initial core turbulence levels. The predictions of the theory reveal several features consistent with observations. The lower the initial turbulence level the higher the amplification of the excitation component will be and hence the higher the spreading rate of the excited jet will be.

Lionel Marraffa, ONERA

I am currently developing a two-dimensional computer code, in association with Professor Dulikravich, from Pennsylvania State University. This code computes internal or external viscous hypersonic flows. It takes into account real gas effects, such as chemical and vibrational non equilibrium.

For such hypersonic flows, some regions of the flow are near chemical equilibrium and chemical time scales are very small compared with aerodynamic ones. If the whole flow was near chemical equilibrium the usual Navier-Stokes codes could be easily adapted to handle the chemistry by simply varying the specific heat ratio γ as a function of pressure P and temperature T, for example using a Mollier chart. But since some regions of the flow can be out of equilibrium or even frozen one usually has to treat the whole domain of computation accounting for non equilibrium chemistry. Thus, the source terms produced by chemical reactions need to be accurately computed; composition is no more a function of P and T only, and becomes an independent variable of the computation.

The computation of these source terms is extremely costly and should be dealt with as accurately as possible, but also very efficiently. Part of my work at Lewis consisted in investigating available methods and alternate approaches applicable to air chemistry. The possibility to generalize the method used for simple dissociation of nitrogen to any number of species was discussed, and would need more work.

Since some of the hypersonic flows of practical interest correspond to expansions of high enthalpy subsonic flows up to hypersonic speeds, I spent also some time working on subsonic inlet conditions for real gas flows.

While working on these two themes, I also continued developing my code, and applied it to hypersonic flows around blunt bodies.

Talks with researchers from Lewis Research Center and with ICOMP visiting scientists have been very helpful for these different activities. Computer and graphic facilities and the persons in charge of these facilities were also very important contributors to the success of my visit. A few new ideas emerged from talks with other specialists and need more testing.

Nessan Mac Giolla Mhuiris, Case Western Reserve University

An attracting set for a dynamical system is a region in phase space which "attracts" nearby initial conditions. Any orbit started in the neighborhood of such a set, A, will evolve towards it and not leave A thereafter. Attracting sets for dissipative systems have dimensions which are less than those of the phase space as a whole and, as they eventually trap all initial conditions, it is their character which governs the long term, asymptotic behavior of the system. Recently, examples have been found of some remarkable attractors. These strange attractors are characterized by the fact that orbits in them, which at some time lie infinitely close together, diverge from each other at an exponential rate and become uncorrelated in a finite time. In effect this precludes quantitative predictions of the behavior of individual orbits for all but the briefest times. Such behavior, termed sensitive dependence on initial conditions, must be present if the system is to be considered chaotic.

Many finite dimensional flows are now known to have this sensitive dependence on initial conditions and it has been conjectured that the appearance of strange attractors is responsible for the evolution of turbulent fluid flows from their laminar precursors. A study is underway to address this conjecture. The rates at which nearby orbits diverge or converge for a given vector field is termed the Lyapunov spectrum of the flow. A strange attractor must have been formulated. This method is capable of measuring both positive and negative exponents. Currently it is being applied to the case of spatially periodic flows in both two and three dimensions. Results obtained for both cases indicate that there are indeed chaotic attractors underlying these flows. The study is being extended to measure the dimension of these sets. An initial report is being prepared for publication.

While a demonstration that the attractor for the Navier-Stokes dynamical system has a finite dimension is interesting, a knowledge of its numerical value does not in any sense describe that set. For example if the dimension turned out to be 10, this does not mean that the entire flow can be captured by the first 10 Fourier modes. Significant breakthroughs in the mathematical description of turbulent behavior will only come if efficient methods are found to capture the essence of the underlying chaotic attractor. A method which shows much promise in this regard is the so-called proper orthogonal decomposition on which I am collaborating with Larry Sirovich of Brown University. We are applying the method here to flow data obtained by numerical simulation of the Navier-Stokes equations with spatially periodic forcing. Using the proper orthogonal decomposition technique it is found that

most of the energy in the flow can be captured by comparatively few modes. We shall project the full partial differential system onto these modes to obtain a low order truncation of the Navier-Stokes equations which still captures the physics to a high degree of accuracy. The low dimensional dynamical system obtained will be studied by means of bifurcation theory, etc.

In another project in collaboration with Charles Speziale of ICASE, we are testing the ability of turbulence models (the k - ε and several second order closures) to accurately predict turbulence statistics for the case of homogeneous turbulent shear flows in a rotating frame. Particular emphasis is being placed on the bifurcations these models undergo as the critical parameter (the ratio of the rotation rate to the shear rate) is varied. Initial comparisons have been made with experimental results from the literature and also with large eddy simulations of the Stanford group. Direct numerical simulations of our own are also being planned to augment these databases. Some reports on this work are being prepared for publication.

Kenneth G. Powell, University of Michigan

My ICOMP-related work this year consisted of two projects. The first project was the development of a solution-adaptive embedded mesh strategy for leading-edge vortex flows. I developed an embedding criterion based on a measure of mesh-convergence, i.e. the quantity $I_h^{2h} U_h - U_{2h}$. This criterion is very general, and should apply to a broader class of problems. In my work, I have shown it to capture vortices and shocks well. I have also developed a way of setting embedding and coarsening thresholds based on:

1) Compiling a histogram of the number of cells flagged for a given threshold;
2) Doing a least squares polynomial fit to give a smooth approximation to the curve;
3) Differentiating twice to find the curvature;
4) Setting the embedding threshold at the highest extremum in the curvature;
5) Setting the coarsening threshold at the lowest extremum in the curvature.

The second project I carried out in conjunction with Bram van Leer. We developed an exciting new algorithm for two-dimensional convection. It is a genuinely two-dimensional upwind scheme of third-order accuracy. It is a cell-vertex scheme in which the residual for a cell is distributed to the two nodes that define the downwind face. We extended the scheme to the Euler equations, making use of the decomposition of Hirsch et al. Preliminary results show outstanding resolution of oblique shocks, and a good deal of promise for optimal convergence rates, but the scheme is far from robust as yet.

Both of these projects are currently being written up as papers for the Reno AIAA meeting, and will be submitted as ICOMP reports.

Majid Rashidi, Cleveland State University

A new computational strategy has been devised to perform Active Vibration Control of rotor systems. Unlike the conventional Active Control strategies in which an elaborate mathematical model of the system as well as the system's modal characteristics are required, here, the vibration attenuation (control) is pursued solely based on the motion characteristics (Vibration Amplitudes) of the system being studied.

This new computational method of vibration attenuation may be called: "A Spectral Optimization Strategy for Active Control of Dynamic Systems". The Second Annual Report - 1987 of ICOMP (NASA TM-100790, page 19) briefly describes the methodology utilized in this new approach as well as its advantages over the conventional approaches.

The vibration attenuation is achieved by minimizing an objective function which is formulated by summing the squares of the vibration of the rotor. The decision variables are the magnitudes and phase angles of sinusoidal force components exerted at different locations along the length of the rotor. The frequencies of these force components are the same as the frequencies of the harmonics of the lateral vibrations (displacements) of the rotor.

Some of the advantages of the new active control strategy over the conventional methods are mentioned in The Second ICOMP Annual Report (page 19). Nevertheless, a potential source of difficulty was encountered, which was circumvented by utilizing proper optimization techniques. It was noticed that a direct search optimization algorithm (one which does not include the derivatives of the objective function) may cause the following problem: although an acceptable final minimum for the objective function is achieved by changing the values of the decision variables, an intermediate magnitude of the objective function, during the optimization process, may become larger than that of the original "to be attenuated" vibration. Of course, in a practical application this means that the system is "kicked" to a worse level of vibration than that of any previous attenuation attempt. This problem was eliminated by utilizing optimization algorithms which include the information regarding the derivatives of the objective function in the search process. Also, imposing proper constraints on the values of the decision variables (magnitudes of the exerted sinusoidal force components) and the value of the objective function, insure a monotonically decreasing envelope of the vibration level of the system.

A rotor system was studied which was modelled by one disk at the center of a flexible shaft supported on two ball bearings. The lateral vibrations of the rotor system were simulated by including three harmonics (1/3 subharmonic, synchronous harmonic, and 3rd superharmonic). The maximum amplitude of the disk vibration was reduced from 35 mils to 6 mils employing the new "Spectral Optimization" approach.

The Spectral Optimization Methods for Active Control of Rotating Machines, as it stands, is suitable for vibration attenuation of a system under a steady state or quasi-steady state condition. The approach has proven to be successful computationally. The next step would be experimental verification of the results produced by the new active control strategy.

Asher A. Rubinstein, Tulane University

The current project is a continuation of the work initiated last summer. The aim of the project is development of fracture mechanics characteristics of materials with complex microstructure. Typically, in these materials, due to internal interaction between a propagating crack and material defects, the crack grows along a curvilinear path. In this project, a typical wavy crack path was modeled as a sinusoidal crack path trajectory. Variation of standard fracture parameters, such as the stress intensity factors and an energy release rate along this trajectory, were investigated. Results demonstrate

that crack growth along a sinusoidal path will require a higher value of the applied load in order to meet a crack growth criteria. The amplitude - wave length ratio is a significant factor in the shielding effect. Thus, even if the crack tip resumes its orientation perpendicular to the tensile stress direction, the shielding effect of the wavy crack path remains in effect and can reduce local value of the energy release rate by 70 per cent.

Avram Sidi, Technion - Israel Institute of Technology

A technical memorandum (ICOMP-88-17) concluding the work that was done in collaboration with M. L. Celestina was completed. This memorandum summarizes the results obtained by applying vector acceleration methods to different 3-D Euler codes.

The possibilities of applying the vector acceleration method to fluid mechanics codes that use multigrid methods and include viscosity were explored in collaboration with W. Usab and J. Yokota. No problems were encountered in accelerating the convergence of a 1-D model problem of W. Usab; however, the 3-D code of J. Yokota seems to need more experimentation. So far it was observed that when applied at a point where the multigrid code starts converging slowly, vector acceleration methods may become effective.

A tutorial series on the development, theory, and practical use of vector acceleration methods was given.

Research on the problem of eigenvalue estimation of linear operators in iterative processes was done, and some very interesting results akin to those that are given for the well known quotient-difference algorithm were obtained.

Patrick Smolinski, University of Pittsburgh

This work involved the development and implementation of parallel processing algorithms for the analysis of transient finite element problems. For the analysis of structural dynamics problems direct time integration methods are the most popular because they can be applied to nonlinear problems. For example, a problem can be nonlinear if the displacements of the structure are large or if the structure is composed of a nonlinear material. With direct integration methods the time period for which the solution is needed is divided into small steps and the solution to the problem is computed at each step. For some nonlinear problems, a new set of equations must be solved at each time step. This can be extremely time consuming for large three-dimensional problems with many thousands of equations. For this reason parallel processing is being investigated to speed-up this type of analysis.

In this study a system of transputer microprocessors was used for the parallel computations. Three different time integration methods, standard explicit time integration, explicit multi-time step integration, and nodal partitioned implicit integration were examined. With explicit methods the displacement at different nodes in the finite element mesh can be computed independently over a time step which allows different parts of the problem to be solved on different processors. Explicit multi-time step integration has this same property and also allows different time steps to be used in different parts of the finite element mesh. This can be useful for problems composed of different materials or which have large differences in element sizes. For implicit time integration, nodal partitioning is used whereby the nodes of the finite element mesh are divided into groups which are updated on different processors. However, for reasonable accuracy with this method the time step is limited by the wave transversal time for the nodal group.

All three algorithms have been used to solve test problems on the transputer system using as many as 40 processors. Significant speed-ups have been obtained through parallel processing with the best speed-ups being achieved in large problems where the communication computation ratio is small. For standard linear structural dynamics problems, the nodal partition implicit method is the most efficient because of its larger stable time step.

Nai-Kuan Tsao, Wayne State University

In this period we have finished the following two reports: 1) On the Accuracy of Solving Triangular Systems in Parallel, and 2) On the Gaussian Elimination and Gauss-Jordan Reduction in Solving Linear Equations.

The first report gives an error complexity analysis of two algorithms for solving a unit diagonal triangular system. The results show that the usual sequential algorithm is optimal in terms of having the minimum maximum and cumulative error complexity measures. The parallel algorithm described by Sameh and Brent is shown to be essentially equivalent to the optimal sequential one. Some numerical experiments are also included.

The second report describes a novel general approach to round-off analysis using the error complexity concept and applies it to the analysis of the Gaussian Elimination and the Gauss-Jordan reduction methods for solving linear equations. The results show that the two algorithms are equivalent in terms of our error complexity measures. Thus the inherently parallel Gauss-Jordan reduction method can be implemented with confidence if parallel computers are available.

Eli Turkel, Tel Aviv University

In addition to my work with Alvin Bayliss, work is also continuing on the properties of central difference schemes coupled with an artificial viscosity. Comparisons of different versions being used in Lewis are in progress. Also the adoption of preconditioners to allow a compressible code to solve low speed flows is being pursued. Applications to problems at NASA Lewis are in progress.

Leon van Dommelen, Florida State University

Currently, the reason why unsteady boundary layers break up and shed their vorticity into the external flow is only understood for flows with symmetries, since these can be computed with high accuracy for given numerical effort. Examples are one-dimensional, two-dimensional and axially symmetric flows. Our study is concerned with uncovering the mechanics of the break up in three-dimensional, arbitrary flows, in order to gain some understanding of the mechanics of flow about complex geometries and, possibly, the processes giving rise to such phenomena as hair-pin vortices inside turbulent boundary layers.

Eulerian methods are presently unable to resolve the separation question even in two dimensions, because of the high resolution needed. In fact the present Eulerian results tend to contradict each other near separation, with only a few exceptions. For that reason, this study chooses a Lagrangian method, which is known to produce results which do agree with the more advanced Eulerian results in two dimensions, is likely to be much more accurate and allows the separation structure to be verified at relatively low resolution. This should continue to hold in three dimensions according to a generalized separation theory developed by van Dommelen and Cowley.

The actual flow chosen is the boundary layer about an ellipsoid with three unequal axes, impulsively set into motion in an arbitrary direction. While the choice of a Lagrangian scheme seems unavoidable to settle the separation question, it does involve some considerable complexities. These arise from the mathematical impossibility to construct a non-singular coordinate system either to use as particle coordinates, or to describe the unknown particle position and velocity.

In this study the first difficulty was resolved by the use of two overlapping meshes, allowing the singular part of each mesh to be computed on the other mesh instead. The second difficulty was resolved by storing all three cartesian position and velocity components, even though they are dependent in the boundary layer limit. Finite difference formulae were created by choosing a local coordinate system for each individual computational point, which allowed a locally non-singular system of difference formulae to be constructed.

During this stay at ICOMP, the program was developed. It was able to reproduce known results in a fully three dimensional form. Currently, the achievable resolution is limited by the memory requirements, with a Cray-XMP CPU-time in the order of 10 minutes at maximum memory.

A considerable further effort is still needed before the question about the unsteady separation process can be regarded as resolved. In particular, (1) repeated computations are needed to identify the relative distributions of mesh points and the stretchings constants which give highest accuracy for given program size; (2) the memory limit should be eliminated by explicitly paging the variables in and out in the form of plates of mesh points as used by the program; (3) various improvements such as insertion of compiler directives to enforce vectorization, faster and more accurate output routines and modified ADI iteration. Such efforts seem well justified in order to understand the basic mechanics of unsteady break up of boundary layers.

Bram van Leer, University of Michigan

The work with M.-S. Liou on the choice of implicit operators for split-flux residuals continued. The paper presented at the 26th Aerospace Sciences Meeting (ICOMP-88-8) was submitted to the Journal of Computational Physics and is in the revision process. It has been pointed out by a reviewer that one of the operators tested, viz. the one based on a particular incomplete linearization of Roe's flux-difference splitting, is not conservative in time, and that this leads to an unfavorable convergence rate or even a limit cycle in the later stages of convergence. At present other linearizations are being tested.

The work with M.-S. Liou and J.-S. Shuen on the numerical treatment of real-gas flow has shifted in emphasis from equilibrium chemistry to finite-rate chemistry. The paper presented at the 1st National Fluid Dynamics Congress (ICOMP-88-7) was submitted to the Journal of Computational Physics and is in the reviewing process.

Pratap Vanka, Argonne National Laboratory

This research is being conducted in collaboration with R. W. Claus of the NASA Lewis Research Center. The objective is to compute complex internal fluid flows efficiently through the use of multigrid techniques. Flows of interest include the interaction of a jet with a cross flow, flow in gas turbine combustors, flow fields in ducts, etc. A finite-volume procedure is used to derive the discrete equations.

In the period from December 5, 1988 to December 22, 1988, a previously developed computer program, MULTIFLOW-3D, was mounted on the NAS CRAY-2 system and three flow situations were calculated with very large grids. In the interactive mode, the transverse jet case of Crabb and Whitelaw was simulated with a grid consisting 48x48x128 (total of 320,000) finite-volumes. This run required about 19 MW of storage and converged results were obtained in about nine minutes of CPU time. The results were plotted with PLOT3D and observed to be in good agreement with expected values. However, it was difficult to establish grid independency as results of a coarser grid were somewhat different than the fine-grid results. Therefore, a much finer grid with 96x96x256 nodes was attempted. This calculation worked fine up to a 3% accuracy limit but then behaved strangely by diverging or oscillating. The cause of this behavior has not yet been identified. We are investigating into possible loss of precision and any other suspicious system problems. In addition, we have calculated a model problem of three-dimensional flow in a shear driven cube with two million nodes at three Reynolds numbers. These runs converged satisfactorily. Also, we performed some preliminary calculations of the flow fields established by a STOVL aircraft in ground proximity. We plan to extend these calculations to more realistic geometries in the future.

J. D. A. Walker, Lehigh University

The work I was involved with at ICOMP this summer was associated with two areas, namely: (1) the development of flows near walls and (2) semi-analytical algorithms for the calculation of nominally steady three-dimensional turbulent flows near walls. Unsteady viscous flows at high Reynolds numbers invariably involve unsteady separation effects where the viscous flow near a wall develops regions of recirculating flow; this is generally followed by focussing of the flow into a very narrow eruptive region and culminates with an ejection of concentrated vorticity from the region near the wall. The immediate interest in the present research is associated with developing algorithms to compute the eruptions that are observed in the production process in turbulent boundary layers; however this type of event occurs in a wide variety of unsteady flows and consequently it is important to develop computational algorithms which can adequately resolve this type of event. In the present research, the evolution and structure of a limit problem for high Reynolds numbers is being considered where an erupting boundary layer is just about to strongly interact with an outer inviscid flow. The computations are being carried out in Lagrangian coordinates wherein the motions of individual fluid particles are tracked as the flow proceeds to interaction.

The second area of research is associated with development of semi-analytical methods to compute three-dimensional turbulent flows near walls and involves a research assistant, Mr. Ammar T. Degani. In this approach analytical profile models, which are based on the coherent structure of the near-wall flow, are used to represent the flow field in the wall layer. A numerical solution is necessary only in the outer part of the flow, where simple turbulence models can be used. Here a numerical solution for the outer flow is matched at any stage in the computations to the wall-layer models. This approach is applicable to Navier-Stokes solutions of the time-mean equation; it is an efficient approach which obviates the need for a large number of mesh points in the near-wall region and at the same time permits accurate evaluation of heat transfer and skin friction coefficients. A number of discussions took place with Lewis personnel relating to the general concepts of this methodology and the implementation of the algorithm in the Lewis PROTEUS code.

David Whitfield, Mississippi State University

These two weeks of August 1988 were spent on a variety of topics ranging from incorporating flux difference splitting residual balancing techniques in some of the rotating machinery codes used at Lewis, to talking to experimentalists to try and determine actual blade-angle setting used in experiments that are being considered for comparison with computational results. Much of the time was devoted to considering how best to utilize modern CFD technology in the areas of acoustics and aeroelasticity.

Yau Shu Wong, University of Alberta

When computing a numerical solution to an unsteady partial differential equation on an infinite domain, it is necessary to perform the calculation on a finite computational domain. A uniform approach to construct absorbing boundary conditions for second-order hyperbolic equations is investigated. The technique is based on the application of the theory of group velocities. Suppose we consider the downstream boundary, such that the outgoing waves have positive group velocities. When reflection occurs, waves with negative group velocities are generated and they are the spurious reflected waves. Thus a condition to eliminate the reflection is to demand that the group velocities be positive at the boundary. To ensure this condition, a pseudo-differential operator is derived. Approximations of the pseudo-differential operator then result in boundary conditions that reduce, but do not totally eliminate, the reflected waves. The effectiveness of the present boundary conditions for linear hyperbolic equations is demonstrated by computational experiments. This research work will be submitted for publication in the Journal of Computational Physics.

Extensions of the absorbing boundary conditions to nonlinear partial differential equations and their application to existing computational fluid dynamics codes (such as the unsteady transonic code ATRAN2 of NASA Ames Research Center) are currently being investigated.

REPORTS AND ABSTRACTS

The ICOMP Steering Committee: "Institute for Computational Mechanics in Propulsion (ICOMP), Second Annual Report - 1987," ICOMP Report No. 88-1, NASA TM-100790, March 1988, 46 pages.

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated by Case Western Reserve University and the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1987.

Hagstrom, Thomas (ICOMP) and Hariharan, S. I. (ICOMP): "Accurate Boundary Conditions for Exterior Problems in Gas Dynamics," ICOMP Report No. 88-2, NASA TM-100807, March 1988, 24 pages.

The numerical solution of exterior problems is typically accomplished by introducing an artificial, far field boundary and solving the equations on a truncated domain. For hyperbolic systems, boundary conditions at this boundary are often derived by imposing a principle of no reflection. However, waves with spherical symmetry in gas dynamics satisfy equations where incoming and outgoing Riemann variables are coupled. This suggests that 'natural' reflections may be important. We propose a reflecting boundary condition based on an asymptotic solution of the far field equations. We obtain nonlinear energy estimates for the truncated problem and present numerical experiments to validate our theory.

Ellis, Graham K. (ICOMP): "Distributed Computation of Graphics Primitives on a Transputer Network," ICOMP Report No. 88-3, NASA TM-100814, prepared for the 1988 Summer Computer Simulation Conference, Seattle, Washington, July 25-28, 1988, 7 pages.

A method is developed for distributing the computation of graphics primitives on a parallel processing network. Off-the-shelf transputer boards are used to perform the graphics transformations and scan-conversion tasks that would normally be assigned to a single transputer based display processor. Each node in the network performs a single graphics primitive computation. Frequently requested tasks can be duplicated on several nodes. The results indicate that the current distribution of commands on the graphics network shows a performance degradation when compared to the graphics display board alone. A change to more computation per node for every communication (perform more complex tasks on each node) may cause the desired increase in throughput.

Ellis, Graham K. (ICOMP): "Two-Dimensional Graphics Tools for a Transputer Based Display Board," ICOMP Report No. 88-4, NASA TM-100820, prepared for the OSU/OCATE and INMOS Parallel Computing Workshop/OCCAM User Group Meeting, Portland, OR, April 11-13, 1988, 16 pages.

A package of two-dimensional graphics routines has been developed in an effort to standardize and simplify the user interface for a transputer based graphics display board. The routines available take advantage of the graphics board's capabilities while also presenting an intuitive approach for generating drawings. The routines allow a user to perform graphics rendering in a two-dimensional real-coordinate space without regard to the actual screen coordinates. Multiple windows, which can be placed arbitrarily on the screen as well as the ability to use double-buffering techniques for smooth animations are also supported. The routines are designed to be run on a transputer other than the graphics display board. The window and screen parameters are maintained locally. The conversion to device coordinates is also performed locally. The only data sent to the display board are control and device coordinate display commands. The routines available include: rotation, translation, and scaling commands; absolute and relative point and line commands; circle, rectangle and polygon commands; and window and viewport definition commands.

Peridier, Vallorie (ICOMP) and Walker, J. David A. (ICOMP): "An Algorithm for Unsteady Flows with Strong Convection," ICOMP Report No. 88-5, NASA TM-100828, October 1988, 14 pages.

An implicit ADI numerical method for the calculation of two-dimensional unsteady flows with strong convection effects is described. The method is based upon the conventional Crank-Nicholson approach for parabolic equations but an upwind-downwind differencing is used for the first order spatial derivatives associated with convection. The differencing is carried out in the current and previous time plane in such a way that the algorithm is second order accurate in both space and time. The difference equations are factored into sequential operators, one in each independent spatial variable; the solution at each time step may then be computed as a sequence of tridiagonal matrix problems. The method may be used in a noniterative manner although iteration at each time step is recommended in situations where the effects of convection are strong.

Dulikravich, George S. (ICOMP); Mortara, Karl W. (Pennsylvania State University); and Marraffa, Lionel (ICOMP): "A Physically Consistent Model for Artificial Dissipation in Transonic Potential Flow Computations," ICOMP Report No. 88-6, NASA TM-100846, May 1988, 14 pages.

The effects that artificial dissipation has on numerical solutions of the transonic Full Potential Equation (FPE) are investigated by comparing the artificial dissipative FPE to a Physically Dissipative Potential (PDP)

equation. Analytic expressions were derived for the variables C and M_c that are used in the artificial density formulation. It was shown that these new values generate artificial dissipation which is equivalent to the physical dissipation which exists in the PDP equation. The new expressions for the variable C and M_c can easily be incorporated into the existing full potential codes which are based either on the artificial density or on the artificial viscosity formulation. A comparison of Physically Dissipative Potential (PDP), Artificial Density or Viscosity (ADV), Artificial Mass Flux (AMF), and ADV with variable C and M_c formulation (MCC) is also presented.

Liou, Meng-Sing (NASA Lewis); Van Leer, Bram (ICOMP); and Shuen, Jian-Shun (Sverdrup): "Splitting of Inviscid Fluxes for Real Gases," ICOMP Report No. 88-7, NASA TM-100856, April 1988, 30 pages.

Flux-vector and flux-difference splittings for the inviscid terms of the compressible flow equations are derived under the assumption of a general equation of state for a real gas in equilibrium. No unnesessary assumptions, approximations or auxiliary quantities are introduced. The formulas derived include several particular cases known for ideal gases and readily apply to curvilinear coordinates. Applications of the formulas in a TVD algorithm to one-dimensional shock-tube and nozzle problems show their quality and robustness.

Liou, Meng-Sing (NASA Lewis) and Van Leer, Bram (ICOMP): "Choice of Implicit and Explicit Operators for the Upwind Differencing Method," ICOMP Report No. 88-8, NASA TM-100857, presented at the AIAA 26th Aerospace Sciences Meeting, Reno, Nevada, January 11-14, 1988, 26 pages.

Flux-vector and flux-difference splittings of Steger-Warming, van Leer and Roe are tested in all possible combinations in the implicit and explicit operators that can be distinguished in implicit relaxation methods for the steady Euler and Navier-Stokes equations. The tests include one-dimensional inviscid nozzle flow, and two-dimensional inviscid and viscous shock reflection. Roe's splitting, as anticipated, is found to uniformly yield the most accurate results. On the other hand, an approximate Roe splitting of the implicit operator (the complete Roe splitting is too complicated for practical use) proves to be the least robust with regard to convergence to the steady state. In this respect, the Steger-Warming splitting is the most robust: it leads to convergence when combined with any of the splittings in the explicit operator, although not necessarily in the most efficient way.

Hon, T.-L. (Lehigh University) and Walker, J. D. A. (ICOMP): "Evolution of Hairpin Vortices in a Shear Flow," ICOMP Report No. 88-9, NASA TM-100858, July 1988, 22 pages.

Recent experimental studies suggest that the hairpin vortex plays an important (and perhaps dominant) role in the dynamics of turbulent flows near walls. In this study a numerical procedure is developed to allow the accurate computation of the trajectory of a three-dimensional vortex having a small core radius. For hairpin vortices which are convected in a shear flow above a wall, the calculated results show that a two-dimensional vortex containing a small three-dimensional disturbance distorts into a complex shape with subsidiary hairpin vortices forming outboard of the original hairpin vortex. As the vortex moves above the wall, it induces unsteady motion in the viscous flow near the wall; numerical solutions suggest that the boundary-layer flow near the wall will ultimately erupt in response to the motion of the hairpin vortex and in the process a secondary hairpin vortex will be created. The computed results agree with recent experimental observations.

Lee, Min Eig (University of Akron); Hariharan, S. I. (ICOMP); and Ida, Nathan (University of Akron): "Solving Time-Dependent Two-Dimensional Eddy Current Problems," ICOMP Report No. 88-10, NASA TM-100875, June 1988, 32 pages.

In this paper we report our results on transient eddy current calculations. For simplicity we consider a two-dimensional transverse magnetic field which is incident on an infinitely long conductor. The conductor is assumed to be a good conductor but not a perfect one. Resulting problem is an interface initial boundary value problem with the boundary of the conductor being the interface. A finite difference method is used to march the solution explicitly in time. The method is shown. Treatment of appropriate radiation conditions are given special consideration. Results are validated with approximate analytic solutions. Two stringent test cases of high and low frequency incident waves are considered to validate the results.

Leonard, B. P. (ICOMP): "Universal Limiter for Transient Interpolation Modeling of the Advective Transport Equations: The ULTIMATE Conservative Difference Scheme," ICOMP Report No. 88-11, NASA TM-100916, September 1988, 116 pages.

A fresh approach is taken to the embarrassingly difficult problem of adequately modeling simple pure advection. An explicit conservative control-volume formulation makes use of a universal limiter for transient interpolation modeling of the advective transport equations. This ULTIMATE conservative difference scheme is applied to unsteady, one-dimensional scalar pure advection at constant velocity, using three critical test profiles: an isolated sine-squared wave, a discontinuous step, and a semi-ellipse. The goal, of course, is to devise a single robust scheme which achieves sharp monotonic resolution of the step

without corrupting the other profiles. The semi-ellipse is particularly challenging because of its combination of sudden and gradual changes in gradient. The ULTIMATE strategy can be applied to explicit conservative schemes of any order of accuracy. Second-order schemes are unsatisfactory, showing steepening and clipping typical of currently popular so-called "high resolution" shock-capturing or TVD schemes. The ULTIMATE third-order upwind scheme is highly satisfactory for most flows of practical importance. Higher order methods give predictably better step resolution, although even-order schemes generate a (monotonic) waviness in the difficult semi-ellipse simulation. But little is to be gained above ULTIMATE fifth-order upwinding which gives results close to the ultimate one might hope for.

Chulya, Abhisak (ICOMP) and Thompson, Robert L. (NASA Lewis): "Improved Finite Strip Mindlin Plate Bending Element Using Assumed Shear Strain Distributions," ICOMP Report 88-12, NASA TM-100928, July 1988, 20 pages.

A linear finite strip plate element based on Mindlin/Reissner plate theory is developed. The analysis is suitable for both thin and thick plates. In the formulation new transverse shear strains are introduced and assumed constant in each two-node linear strip. The element stiffness matrix is explicitly formulated for efficient computation and computer implementation. Numerical results showing the efficiency and predictive capability of the element for the analysis of plates are presented for different support and loading conditions and a wide range of thicknesses. No sign of shear locking phenomenon was observed with the newly developed element.

Ellis, Graham K. (ICOMP): "User's Manual for the Two-Dimensional Transputer Graphics Toolkit," ICOMP Report 88-13, NASA TM-100974, August 1988, 104 pages.

The user manual for the two-dimensional graphics toolkit for a transputer based parallel processor is presented. The toolkit consists of a package of two-dimensional display routines that can be used for simulation visualizations. It supports multiple windows, double buffered screens for animations, and simple graphics transformations such as translation, rotation, and scaling. The display routines are written in occam to take advantage of the multiprocessing features available on transputers. The package is designed to run on a transputer separate from the graphics board.

Ellis, Graham K. (ICOMP): "Implementing Direct, Spatially Isolated Problems on Transputer Networks," ICOMP Report 88-14, NASA TM-101297, August 1988, 58 pages.

Parametric studies have been performed on transputer networks of up to 40 processors to determine how to implement and maximize the performance of the solution of problems where no processor-to-processor data transfer is

required for the problem solution (spatially isolated). Two types of problems were investigated in this study. A computationally intensive problem where the solution required the transmission of 160 bytes of data through the parallel network, and a communication intensive example that required the transmission of 3 Mbytes of data through the network. This data consists of solutions being sent back to the host processor and not intermediate results for another processor to work on. Studies were performed on both integer and floating-point math unit and offers approximately an order of magnitude performance increase over the integer transputer on real valued computations. The results indicate that a minimum amount of work is required on each node per communication to achieve high network speedups (efficiencies). The floating-point processor requires approximately an order of magnitude more work per communication than the integer processor because of the floating-point unit's increased computing capability.

Choo, Yung K. (NASA Lewis); Eiseman, Peter R. (ICOMP); and Reno, Charles (NASA Lewis): "Interactive Grid Generation for Turbomachinery Flow Field Simulations," ICOMP Report No. 88-15, NASA TM-101301, December 1988, 9 pages.

The control point form of algebraic grid generation presented here provides the means that are needed to generate well-structured grids for turbomachinery flow simulations. It uses a sparse collection of control points distributed over the flow domain. The shape and position of coordinate curves can be adjusted from these control points while the grid conforms precisely to all boundaries. An interactive program called TURBO, which uses the control point form, is being developed. Basic features of the code are discussed and sample grids are presented. A finite-volume LU implicit scheme is used to simulate flow in a turbine cascade on grid generated by the program.

Jiang, Bo-Nan (ICOMP) and Chang, C. L. (Cleveland State U.): "Least-Squares Finite Elements for Stokes Problem," ICOMP Report No. 88-16, NASA TM-101308, December 1988.

A least-squares method based on the first-order velocity-pressure-vorticity formulation for the Stokes problem is proposed. This method leads to a minimization problem rather than a saddle-point problem. The choice of the combinations of elements is thus not subject to the Ladyzhenskaya-Babuska-Brezzi (LBB) condition. Numerical results are given for the optimal rate of convergence for equal-order interpolations.

Sidi, Avram (ICOMP) and Celestina, Mark L. (Sverdrup): "Convergence Acceleration for Vector Sequences and Applications to Computational Fluid Dynamics," ICOMP Report No. 88-17, NASA TM-101327, August 1988, 24 pages.

Some recent developments in acceleration of convergence methods for vector sequences are reviewed. The methods considered are the minimal polynomial extrapolation, the reduced rank extrapolation, and the modified minimal polynomial extrapolation. The vector sequences to be accelerated are those that are obtained from the iterative solution of linear or nonlinear systems of equations. The convergence and stability properties of these methods as well as different ways of numerical implementation are discussed in detail. Based on the convergence and stability results, strategies that are useful in practical applications are suggested. Two applications to computational fluid mechanics involving the three-dimensional Euler equations for ducted and external flows are considered. The numerical results demonstrate the usefulness of the methods in accelerating the convergence of the time-marching techniques in the solution of steady-state problems.

Chang, Shih-Hung (ICOMP) and Liou, Meng-Sing (NASA Lewis): "A Numerical Study of ENO and TVD Schemes for Shock Capturing," ICOMP Report No. 88-18, NASA TM-101355, September 1988, 22 pages.

The numerical performance of a second-order upwind-based TVD scheme and that of a uniform second-order ENO scheme for shock capturing are compared. The TVD scheme used in this study is a modified version of Liou, using the flux-difference splitting (FDS) of Roe and his "superbee" function as the limiter. The construction of the basic ENO scheme is based on Harten, Engquist, Osher, and Chakravarthy, and the 2D extensions are obtained by using a Strang-type of fractional-step time-splitting method. Numerical results presented include both steady and unsteady, 1D and 2D calculations. All the chosen test problems have exact solutions so that numerical performance can be measured by comparing the computed results to them. For 1D calculations, the standard shock-tube problems of Sod and Lax are chosen. A very strong shock-tube problems, with the initial density ratio of 400 to 1 and pressure ratio of 500 to 1, is also used to study the behavior of the two schemes. For 2D calculations, the shock wave reflection problems are adopted for testing. The cases presented in this report include flows with Mach numbers of 2.9, 5.0 and 10.0.

Tsao, Nai-Kuan (ICOMP): "On the Accuracy of Solving Triangular Systems in Parallel," ICOMP Report No. 88-19, NASA TM-101384, November 1988, 26 pages.

An error complexity analysis of two algorithms for solving a unit-diagonal triangular system is given. The results show that the usual sequential algorithms is optimal in terms of having the minimal maximum and cumulative error complexity measures. The parallel algorithm described by Sameh and Brent is shown to be essentially equivalent to the optimal sequential one. Some numerical experiments are also included.

Kim, S.-W. (ICOMP): "A Near-Wall Turbulence Model and Its Application to Fully Developed Turbulent Channel and Pipe Flows," ICOMP Report No. 88-20, NASA TM-101399, November 1988, 30 pages.

A near-wall turbulence model and its incorporation into a multiple-time-scale turbulence model are presented in this report. In the method, the conservation of mass, momentum, and the turbulent kinetic energy equations are integrated up to the wall; and the energy transfer rate and the dissipation rate inside the near-wall layer are obtained from algebraic equations. The algebraic equations for the energy transfer rate and the dissipation rate inside the near-wall layer have been obtained from a k-equation turbulence model and the near wall analysis. A fully developed turbulent channel flow and fully developed turbulent pipe flows were solved using a finite element method to test the predictive capability of the turbulence model. The computational results compared favorably with experimental data. It is also shown that the present turbulence model could resolve the over-shoot phenomena of the turbulent kinetic energy and the dissipation rate in the region very close to the wall.

Mankbadi, Reda R. (ICOMP); Raman, Ganesh (Sverdrup); and Rice, Edward J. (NASA Lewis): "Effects of Core Turbulence on Jet Excitability," ICOMP Report No. 88-21, NASA TM-101405, AIAA-89-0966.

The effects of varying freestream core turbulence on the evolution of a circular jet with and without tonal excitation are examined. Measurements are made on a 8.8 cm diameter jet at a Mach number of 0.3. The jet is excitated by plane waves at a Strouhal number of 0.5. For the excited and unexcited cases the turbulence level is varied by screens and grids placed upstream of the nozzle exit. The experimental results are compared with a theoretical model which incorporates a variable core turbulence and considers the energy interactions between the mean flow, the turbulence and the forced component. Both the data and the theory indicate that increasing the freestream turbulence diminishes the excitability of the jet and reduces the effect of excitation on the spreading rate of the jet.

Duck, Peter W. (ICOMP): "Three-Dimensional Marginal Separation," ICOMP Report No. 88-22, NASA TM-101411, December 1988, 32 pages.

The three-dimensional marginal separation of a boundary layer along a line of symmetry is considered. The key equation governing the displacement function is derived, and found to be a non-linear integral equation in two space variables. This is solved iteratively using a pseudo-spectral approach, based partly in double Fourier space, and partly in physical space. Qualitatively the results are similar to previously reported two-dimensional results (which are also computed to test the accuracy of the numerical scheme); however quantitatively the three-dimensional results are much different.

SEMINARS

Dare Afolabi, Purdue University at Indianapolis: "Effects of Inter-Stage Coupling on the Vibration of Mistuned Bladed Disks"

The rotating stage of a turbomachine is usually known as a bladed disk. In previous vibration analysis of bladed disks by various workers, it has been customary to consider only one stage in isolation. In an actual engine, several bladed disks are interconnected through the residual flexibility of the shaft. In this presentation, the combined effects of mistuning and inter-stage coupling on the vibration characteristics of turbine blades is discussed. It is shown that the characteristic double modes of a tuned single-stage are split into a cluster of several modes within a narrow spectral band. The high modal density resulting from such splitting leads to a considerable interference among the system modes, to an extent much more complicated than the case of an isolated bladed disk. However, in order to account for the inter-stage coupling of every blade in an engine, considerable computational resources are required. A concurrent computational strategy using parallel computers is proposed.

Suresh K. Aggarwal, University of Illinois at Chicago: "Vaporization and Ignition Characteristics of Multicomponent Fuel Sprays"

Recent work on the vaporization and ignition characteristics of multi-component fuel sprays will be presented. Emphasis will be on the effects of transient liquid-phase processes on the spray behavior and on the comparison of three global reaction mechanisms for the oxidation of hydrocarbon fuels. Finally the concept of using multicomponent fuels for enhancing the ignitability of sprays will be discussed.

A. J. Baker, University of Tennessee, Knoxville: "Recent Advances in Finite Element CFD Analysis"

This seminar will highlight recent developments on a class of finite element CFD algorithms for application to fluid mechanics problems of pertinence. The theoretical basis is the classic Galerkin weak statement, hence the fundamental decision of trial (and test) space is clearly introduced. The analysis for stability, hence solution accuracy, is introduced in a natural way using an evolutionary Taylor series. The end point is derivation of a modified form of the Navier-Stokes conservation law system. The resultant Taylor weak statement (TWS) thus introduces, in a nondiscretized statement form, each fundamental decision required of an algorithm designer. The results of definitive theoretical analyses are summarized, and numerical results for a variety of pertinent problem definitions will be summarized to document performance.

Alvin Bayliss, Northwestern University: "An Adaptive Pseudo-Spectral Method with Application to Problems in Combustion"

An adaptive pseudo-spectral procedure is developed and applied to several problems in the combustion of both gaseous and solid fuel. The numerical procedure is based on introducing a family of coordinate transformations to transform the original problem. The optimal coordinate system is chosen dynamically so as to minimize a certain functional of the solution which measures the spectral interpolation error. The method has been shown to be effective for the rapidly varying functions occurring in combustion.

The numerical method has been used to investigate higher order instabilities arising in certain combustion models. In solid fuel combustion we compute an oscillatory solution initially predicted by analysis. The computations show that the initially sinusoidal oscillation develops into relaxation oscillations as a certain parameter is increased. A period doubling secondary bifurcation is computed beyond a critical parameter value. Upon further increasing the parameter we find that the period doubled solution can no longer be computed and the solution becomes to singly periodic again.

In gaseous combustion we consider the problem of a flame stabilized by a line source of fuel. As the strength of the fuel source increases, stable, steady state calls develop as predicted by analysis. As this parameter is increased further we find a sequence of subcritical bifurcations leading to bistable cells of increasing mode number.

Seo Won Choi, Stanford University: "Steady Fully-Developed Flow Through a Slowly-Twisted Elliptic Pipe"

Torsion effects in curved-pipe flows are studied by examining the fully-developed steady laminar flow in a slowly-twisted pipe with a constant elliptic cross-section. The centerline of the pipe is straight and the cross-section is rotated about the centerline in axial direction, so that the flow exhibits torsion effects not associated with curvature of the centerline. A series solution is found up to second order in closed form with the aid of symbolic manipulation, in which the dimensionless torsion of the centerline is the small parameter ε . The coefficients are rational fractions and uniformly valid as functions of two parameters: Reynolds number and aspect ratio.

Effects of torsion on transverse flow are categorized into two different kinds based on the parameters involved. The primary effect is of first order and a result of an interaction between two parameters, aspect ratio (its departure from unity) and torsion, so it is kinematic. The secondary effect of torsion is an inertia effect which is determined by the Reynolds number and the dimensionless torsion. The inertia effect is of second order and a effect of torsion in a steady fully-developed flow through a helically-coiled pipe of circular cross-section. The existence of the displacement effect indicates that the torsion effect on a helically-coiled-pipe flow will be of first order and comparable to the curvature effect if the pipe cross-section is non-circular.

Stephen Cowley, Imperial College, London: "On the Instability of Hypersonic Flow Past a Wedge, An Informal Seminar"

The instability of a compressible flow past a wedge is investigated in the hypersonic limit. Particular attention is given to Tollmien-Schlichting waves governed by triple-deck theory through some discussion of inviscid modes is given. It is shown that the attached shock has a significant effect on the growth rates of Tollmien-Schlichting waves. Moreover, the presence of the shock allows for more than one unstable Tollmien-Schlicting wave. Indeed an infinite discrete spectrum of unstable waves is induced by the shock, but these modes are unstable over relatively small but high frequency ranges. The shock is shown to have little effect on the inviscid modes considered by previous authors and an asymptotic description of inviscid modes in the hypersonic limit is given.

William N. Dawes, Whittle Laboratory, Cambridge University, U.K.: "A Navier-Stokes Solver as a Teaching Aid for 3D Turbomachinery Flows"

A simple, robust 3D Navier-Stokes solver will be described and its use in understanding some of the complex flows in turbomachinery will be illustrated by applications to a variety of configurations.

Peter W. Duck, University of Manchester, U.K.: "Unsteady Triple-Decks: Boundary-Layer Instability and Transition"

Triple-deck theory was originally developed in order to better explain and understand the separation of boundary layers, and has since been applied to a vast number of diverse situations. Amongst these has been the description of the infinite Reynolds number limit on the Tollmien-Schlichting mode (lower branch) of instability of Blasius type boundary layers. More recently the theory has formed the basis for describing (rationally) the vexing problem of the early stages of certain boundary-layer transition processes, with some encouraging results.

One of the key questions appears to be the role non-linearity plays in these processes. A number of computations based on the non-linear triple-deck will be presented, some of which exhibit a finite-time breakdown, which will be discussed; we speculate as to the physical significance of this, which is conceivably linked to a burst of vorticity into the main body of the fluid, related to an early stage in some boundary-layer transition process.

Recent results obtained regarding linear stability (based on the triple deck) will also be discussed, in particular pertaining to supersonic flows of the axisymmetric and 3D class.

George S. Dulikravich, Pennsylvania State University: "A Seminar in Two Parts"

Part I. Iterative Acceleration and Physically Based Dissipation for Euler Equations of Gasdynamics. A new algorithm for the acceleration of explicit iterative schemes for a system of partial differential equations has been developed. The method is based on the idea of allowing each partial differential equation in the system to approach the converged solution at its own optimal speed. The DNLMR (Distributed Non-Linear Minimal Residual) method allows a separate sequence of optimal weighting factors to be used for each component of the general solution vector. The acceleration scheme was applied to the system of time-dependent Euler equations of inviscid gasdynamics in conjunction with the finite volume Runge-Kutta explicit time-stepping method with the Jameson's Artificial Dissipation (AD) terms and the newly formulated Physically Based Dissipation (PBD) model. The PBD model uses physical dissipation terms from the Navier-Stokes equations of gasdynamics, while enforcing slip boundary conditions of inviscid gasdynamics and utilizing spatially varying viscosity coefficients. Tests were performed for various flow conditions, including internal flow, flow around a cylinder and flow over an airfoil with AD and PBD. Using DNLMR, between 30% and 60% of the computational efforts were saved in the subsonic compressible flow calculations.

Part II. A Comparative Study of Iterative Algorithms for the Euler Equations of Gasdynamics. A comparative study for the solution of the Euler equations has been performed using four Flux-Vector-Splitting (FVS) schemes and a central difference scheme with two different dissipation models. All schemes were tested for the case of steady, inviscid, transonic airfoil flow. The dissipation models were used to calculate subsonic and transonic flow around a circular cylinder Van Leer's FVS scheme was formed to be robust and appears to generate little numerical dissipation. The FVS schemes of Deese and Steger yield results similar to Van Leer's, though not quite as robust. Whitfield's FVS scheme generates large amounts of numerical dissipation and causes delayed post-shock pressure recovery. A new, Physically Based Dissipation (PBD) model for central difference schemes has been developed and compared to Jameson's Artificial Dissipation (AD) model. The PBD model requires only one user specified coefficient, does not alter the continuity equation like the AD model, and does not require fourth order dissipation terms. The PBD model yields sharp shock and exhibits consistent robustness over a wide range of Mach numbers. Several difference viscosity sensors have been suggested and tested indicating that a sensor based on the divergence of the velocity vector may yield better results than pressure sensors.

Peter R. Eiseman, Columbia University: "A Continuation of the Tutorial on Algebraic Grid Generation"

In the first tutorial, algebraic grid generation was thoroughly examined for the case of construction in a single direction. This culminated in the establishment of the general multi-surface transformation together with various interpolation schemes. In the present continuation, we bring our

attention to bear upon the appropriate means to assemble directional constructions. This will quite naturally lead to a control point form of algebraic grid generation. As in the previous tutorial, our development will be paced by the attendees rather than by a set amount of material

Mohammad Farshchi, Nielsen Engineering & Research, Inc.: "A Probability Density Function Closure Model for Compressible Turbulent Chemically Reacting Flows"

The objective of this work is to develop closure models for compressible turbulent reacting flow fields with large velocity and thermo-chemical fluctuations. The application of a stochastic joint probability density function (PDF) model provides a consistent method of inclusion of exact finite rate chemistry kinetics. This model treats the chemical reaction source terms and state equations exactly. Other thermo-chemical models and their limitations are also discussed. Any level of thermo-chemical closure model requires an understanding and modeling of coupling between vorticity, entropy and acoustic modes in the nonreacting compressible turbulent flows. To study the effect of compressibility on turbulent flows a transport equation for a single point joint PDF for velocity, pressure, and entropy is developed and analyzed for insight to such modeling.

J. S. B. Gajjar, Exeter University: "Upper-Branch Stability of Compressible Boundary Layer Flows"

In this work we extend some of the ideas of incompressible stability theory to compressible flows and we consider scalings relevant to the upper-branch of two-dimensional compressible boundary layer flows with and without heat transfer. For pressure gradient boundary layer flows the significant changes due to compressibility arise for O(1) Mach numbers.

Nonlinear effects are also included and this brings into play the ideas of nonlinear compressible critical layers. For Blasius flow with insulated walls the major changes due to compressibility occur for small Mach numbers.
In both cases we show how the properties of neutral inflexional waves are retrieved.

J. S. B. Gajjar, Exeter University: "Stationary and Non-Stationary Cross-Flow Vortices in Three-Dimensional Boundary Layer Flow"

In this talk we consider the upper-branch neutral stability of three-dimensional distrubances imposed on a three-dimensional boundary layer profile, and in particular we investigate stationary and non-stationary cross-flow vortices. Cross-flow vortices with small wavespeeds have a disturbance structure analogous to upper-branch two-dimensional Tollmien-Schlichting waves and we show how the linear and nonlinear eigenrelations can

be obtained. For O(1) wavespeeds the cross-flow vortices have a disturbance structure first discussed by Gregory-Stuart and Walker (1955). We extend this work also to include nonlinear effects and some of the results and special limiting cases will be discussed.

Peyman Givi, State University of New York at Buffalo: "Compositional Structure of a Quenched Unpremixed Flame: Spectral-Element Simulations"

The spectral-element method, a numerical scheme that combines the accuracy of spectral methods with the versatility of finite element techniques, has been employed to study the mechanisms of mixing and chemical reactions in an unpremixed flame stabilized on a two-stream planar mixing layer. The results of simulations of the harmonically forced, spatially developing flow are statistically analyzed to examine the compositional structure of the flame near quenching. The results indicate that as the flame approaches extinction, the mean and the rms values of the reactant concentration decrease, while those of the product concentration and temperature increase. This behavior is enhanced by increasing the hydrodynamic characteristic time (reducing the local Damkohler number) and is consistent with that observed experimentally.

Thomas Hagstrom, State University of New York at Stony Brook: "Asymptotic Boundary Conditions for Computational Fluid Dynamics"

A variety of basic problems in theoretical fluid dynamics are posed in unbounded spatial domains. Computational simulations of such problems naturally require truncation of the physical domain and the imposition of boundary conditions at artificial boundaries. The choice of these conditions has attracted much interest and controversy. Our approach is to use far-field asymptotics to derive accurate conditions. The difficulties and consequences of such an approach highlight the contrasting physical settings: compressible -- incompressible, viscous -- inviscid, steady -- unsteady, exterior -- wall-bounded. Results are presented for the Euler equations in an exterior domain and the incompressible Navier-Stokes equations in a channel.

S. I. Hariharan, University of Akron: "Time Dependent Calculations in Electromagnetic Wave Propagation"

This talk will consist of discussions on electromagnetic wave propagation in the situation of eddy current calculations. While the problem is linear, the presence of certain small and large parameters (simultaneously) make the computational aspects of the problem difficult. In addition boundary conditions for the problem are hard to handle numerically. These difficulties will be discussed and our results will be shown. Some open questions will also be discussed.

S. I. Hariharan, University of Akron: "Time Dependent Calculations in Acoustic Wave Propagation"

This talk will discuss transient calculations of acoustic wave propagation in the atmosphere. Difficulties arise due to boundary conditions. The problem here is nonlinear. Moreover it is in a nonconservative form. To solve it numerically, an asymptotic expansion is used to reduce the nonlinear problem into a sequence of linear problems. Those linear problems can be cast in a conservation form. Combination of solution of the first two linear problems are shown to yield the solution of the nonlinear problems. Numerical results will be presented.

Giussepe S. Iannelli, University of Tennessee: "Finite Element Methods for Hyperbolic Conservation Law Systems"

A finite element method is developed to simulate predominantly convective fluid flow processes. This method synergistically utilizes three procedures: 1) a Taylor weak statement, which provides for derivation of companion conservation law systems with embedded dispersion error control mechanisms, 2) a highly stable 2nd-order accurate stiffly-stable implicit Runge-Kutta temporal algorithm, a potential improvement over the conventional backward Euler and Crank-Nicolson (trapezoidal) rules, and 3) a matrix tensor product factorization that permits efficient handling of the final large linear algebra problem.

These procedures have been implemented within a finite element semi-discrete framework in a structured code embodiment. Numerical solutions have been generated and compared to conventional algorithm solutions for: 1) Unsteady quasi-one dimensional Euler predictions with shocks; 2) A specially formulated 2-D hyperbolic conservation law system with available analytical solution; 3) Two-dimensional depth-averaged free surface hydromechanical flow and gasdynamic Euler equations.

Bo-Nan Jiang, University of Texas, Austin: "Least-Squares Finite Element Method for First-Order System of Partial Differential Equations"

A least-squares finite element method for linear and nonlinear first-order systems of partial differential equation in engineering and mathematical physics is developed. The procedure is implemented in conjunction with an element-by-element preconditioned conjugate gradient method. An adaptive refinement strategy is developed. The theory and method can be used to develop a general purpose finite element program for problems in fluid dynamics and solid mechanics.

Bo-Nan Jiang, University of Texas, Austin: "Least-Squares Finite Element Method for Fluid Dynamics"

More than two decades ago people started to apply the finite element method to fluid dynamics problems. A large number of papers have been published, but at present only two commercial finite element codes for incompressible flow are available. One reason is that there was no unified method which could cover a wide range of fluid dynamics problems. People use a particular method to solve a particular problem. It is thus difficult to implement these methods in a general-purpose code.

This talk will introduce a unified least-squares method which can be the basis for a general-purpose code to solve many problems not only in fluid dynamics but also in solid mechanics. The least-squares method will be compared with current commonly used finite element methods. Numerical results for various problems from the incompressible Navier-Stokes equations to the compressible Euler equations will be presented.

S.-W. Kim, ICOMP: "A Multiple-Time Scale Turbulence Model"

A multiple-time-scale turbulence model of a single point closure and a simplified split-spectrum method is presented. Use of the variable partitioning method in the multiple-time-scale turbulence model yielded a few improvements over a class of single-time-scale turbulence models. These improvements include: the capability to model the cascade of turbulent kinetic energy; the capability to model the effect of the ratio of production rate to the dissipation rate on eddy viscosity; and the capability to resolve the viscous super layer, among many others.

Example problems considered are: a fully developed channel flow, a plane jet exhausting into a moving stream, a wall jet flow, a wake-boundary layer interaction flow, a turbulent backward-facing step flow, a confined coaxial jet, and a confined coaxial swirling jet. The computational results compared favorably with experimental data for the example problems considered. For complex turbulent flows, such as the wall jet flow and the confined coaxial jets, the present turbulence model yielded significantly improved computational results compared with the available computational results obtained by using the single-time-scale turbulence models such as the two-equation turbulence models and the Reynolds stress turbulence models.

S.-W. Kim, ICOMP: "A Control-Volume Based Reynolds Averaged Navier-Stokes Equation Solver Valid at all Flow Velocities"

A control-volume based finite difference method valid at all flow velocities is presented. In the methods, all flow variables are solved in their primitive forms and the pressure is updated by solving a convection-diffusion equation for incremental pressure.

The turbulent flows were solved using a k-ε turbulence model supplemented with a near-wall turbulence model. The turbulence model can be classified as a partially low Reynolds number k-ε turbulence model to distinguish it from

the conventional two-layer turbulence models and low Reynolds number turbulence models. It is shown that the turbulence model can resolve details of the near-wall turbulence structure, and that significantly improved computational results have been obtained for all flow cases considered.

The example problems considered include: (1) developing channel flow; (2) developing pipe flow; (3) flow through a 90 degree bend in a two-dimensional duct; (4) polar cavity flow at Reynolds numbers 60 and 350; (5) incompressible turbulent flow over a curved hill at Reynolds number of 1.0×10^7; (6) supersonic turbulent flow (Mach number of 3.5) over a compression ramp; and (7) shockwave-turbulent boundary layer interaction in a transonic flow over an axisymmetric curved hill.

Limitations of the existing numerical methods as well as turbulence models as applied to the above example problems are summarized as follows:

1) For the incompressible turbulent flow over a curved hill, the flow recirculation zone at the rear end of the hill has not been captured as yet even with the use of the curvature correction term included in the turbulence equations;

2) For the flow over a compression corner, the correct shape of the shockwave induced flow recirculation zone has seldom been captured in numerical calculations; and

3) For the transonic flow over an axisymmetric curved hill, prediction of the correct location of the shock wave still remains as unsolved.

It is shown that the present numerical method, when used with the partially low Reynolds number $k-\varepsilon$ turbulence model, yielded sufficiently accurate computational results which are free of these shortcomings.

W. Kollmann, University of California, Davis: "Pdf Methods for Turbulent Flows"

Pdf methods for turbulent flows with chemical reactions will be discussed. The calculation of non-premixed turbulent flames burning hydrogen with air using several chemical models will illustrate the approach.

Sang Soo Lee, Brown University: "Multiple Coherent Mode Interaction in a Developing Round Jet"

The integral energy method has been used in order to study the large-scale coherent wave mode interactions in a spatially developing round jet of a large Reynolds number. A flow quantity is split into two components; time-independent mean flow and large-scale coherent structure. The large-scale structure is decomposed into three fundamental wave modes (axisymmetric, first-order and second order helical) and two subharmonic modes (axisymmetric and first-order helical). The streamwise development of a jet is obtained in terms of the mean flow shear layer momentum thickness, the wave mode kinetic energy and the wave mode phase angle. The results of the five mode interaction show that the axisymmetric and helical modes grow almost identically in the initial region until the energy densities of the fundamental modes reach peak values. When the initial energy densities of the fundamental and subharmonic modes are equal, the initial growth of the shear layer momentum thickness and the fundamental energy densities is mainly

governed by the energy transfer from the mean flow to the fundamental modes. The subharmonic energy densities reach peak values at earlier or later streamwise positions than in the decoupled case depending upon the nonlinear mode-mode energy transfers. The nonlinear interaction between wave modes is strongly dependent on the phase angle differences between wave modes as well as the initial energy densities play a significant role in the streamwise evolution of the large-scale coherent wave modes and mean flow.

Avi Lin, Temple University: "Parallel Computational Fluid Dynamics on Distributed Systems"

The main goal of this paper is to show theoretical results and to discuss the nature of the distributed parallel algorithms to computational fluid dynamics (CFD) with applications to two- and three-dimensional incompressible steady state flow fields. This general implicit stable iterative approach exhibits a near optimal performance for these fields and enjoys several important features: (1) for large numerical fields, the design of the appropriate parallel algorithm is insensitive to the number of processors, and, in fact, its performance grows monotonically with them; (2) it works especially good for large numerical fields, with dimensions larger than the number of processors in the system, and in this case it achieves an optimal speed up and a very low communication complexity; (3) this family of the parallel CFD algorithms can be mapped onto any parallel MIMD architecture without major programming difficulties, or algorithmical changes.

Avi Lin, Temple University: "Experience, Thoughts and Afterthoughts for Some Recent Turbulence Models"

The "two-equations" models, and especially the "k-ε" model are very common, and widely used in the CFD community. We will discuss the mathematical nature of this class of model, and what kind of performance should be expected. We will discuss the sensitivity of the coefficients to different classes of flows and will point out the major drawbacks of these models. As a result we will propose another way of enlarging the class of flows for which Reynolds stress models can be applied.

Jenz Lorenz, California Institute of Technology: "Attractors for Navier-Stokes Equations"

In two space dimensions, the time-dependent Navier-Stokes equations have a universal attractor, the "size" of which depends on the Reynolds number. For three space dimensions, the corresponding result is known only under additional assumptions. The size of the attractor is related to the smallest scale in the problem, a quantity of interest for computations.

Reda R. Mankbadi, Cairo University: "On the Conditions for Resonance Interactions of Instability Waves in the Axisymmetric Jet"

The conditions for resonance interaction between two instability waves in an axisymmetric jet were investigated. Considerations of the energy equation of the wave resulting from the interaction indicate that the phase angle between the wave-induced stresses and the wave-induced strains plays a crucial role in the resonance interaction. This fact is demonstrated experimentally by exciting a jet at fundamental and subharmonic frequencies. The phase angle between the waves' stresses and strains was varied by varying the initial phase-difference between the two excitation waves. The subharmonic resonance was found to be highly dependent on this angle. Favorable agreement was found between the phase angle predicted by a nonlinear theory and the measurements.

Lionel Marraffa, ONERA, France: "Numerical Simulation of Two Dimensional Viscous Flows with Real Gas Effects"

A computer code computing two dimensional hypersonic viscous flows is currently under development. This program is able to treat internal or external flows, in cartesian or cylindrical (axisymmetric) coordinates. It takes into account real gas effects such as finite rate chemistry and vibrational non equlibrium. This code uses an explicit MacCormack scheme, coupled with an implicit calculation of chemical and vibrational source terms. After a presentation of the model and of the method used, preliminary results will be presented for simple reaction-convection- diffusion processes, for low speed flows.

John J. H. Miller, Trinity College, Dublin, Ireland: "Mixed Finite Element Methods for Stiff Systems of Non-Linear Parabolic and Elliptic Equations"

We discuss discretization methods for a system of nonlinear parabolic and elliptic equations in a conservation form. We are interested mainly in space discretizations in three dimensions. Special consideration is given to the construction of appropriate discretizations in the presence of singular perturbations. The main technique is a non-standard mixed finite element method.

Kenneth G. Powell, University of Michigan: "Modeling Leading Edge Vortex Flows with an Embedded Mesh Procedure"

Leading-edge vortices are an important and interesting aerodynamic phenomenon which occur on highly swept wings at moderate to high angles of attack. The modeling of these flows using a finite-volume formulation of the Euler equations will be discussed. The approach uses regions of local mesh refinement which provide very high-resolution solutions at a low computational cost. The approach will be outlined, and results of the scheme presented. An important issue is the role of artificial viscosity versus real viscosity in the establishment of the vortical flow and the accompanying total pressure

loss. A similarity solution for a high Reynolds number approximation to the Navier-Stokes equations that helps to explain the computational results will be presented. Solutions will be compared with experimental data.

Stanely G. Rubin, University of Cincinnati: "RNS Formulations for Viscous Interacting Flow"

A composite Euler/boundary layer system of equations formed reduced Navier-Stokes or RNS is applied for the computation of incompressible to supersonic flow. For the former, a form of flux-vector splitting leads To a pressure relaxation procedure and for the latter a multiplicative composite pseudo-potential/vortical velocity leads to a 'potential' relaxation procedure. Solutions are described for a variety of two- and three-dimensional flows with confined regions of recirculation. A form of laminar flow breakdown is discussed.

Asher A. Rubinstein, Tulane University: "Mechanics of Crack Path Formation"

Two aspects of curvilinear crack path trajectory are investigated. The first aspect deals with the toughening mechanism based on the crack path deflection. The second part of the investigation is aimed at the mechanics of crack kinking and curving. The effect of material toughening is evaluated by comparing the remote stress field parameters, such as the stress intensity factors (controlled by a loading on a macroscale), to effective values of these parameters acting in the vicinity of a crack tip (microscale). The effects of the curvilinear crack path are separated into three groups: crack tip direction, crack tip geometry pattern shielding, and crack path length change.

Crack path formation mechanics is studied by analyzing experimentally obtained crack path trajectories in a heterogeneous stress field. The experimental crack path trajectories were used as data for the numerical simulations, recreating the actual stress field governing the development of the crack path. Thus, the current theories of crack curving and kinking could be examined by comparing them with the actual stress field parameters as they develop along the experimentally observed crack path. The numerical simulation, based on the solution of equivalent boundary value problems with possible perturbations of the crack path is presented.

Ashraf Saad-El-Din Sabry, Brown University: "Numerical Simulation of the Nonlinear Evolution of Gortler Vortices"

A numerical simulation of the nonlinear temporal evolution of Gortler vortices is performed. The model is able to recover both qualitatively and quantitatively the most recent experimental observations. Similar flow structures are observed and extra flow quantities are computed to enhance our physical understanding of the problem. The computation showed that the streamwise vortices themselves do not undergo transition directly, but instead, generate a localized intense free shear layer that ultimately breaks down into the turbulence. This has been demonstrated by the computation of the total vorticity which showed the significant effect of the streamwise vortices in ejecting from the wall a considerable bulk of fluid characterized

by an intense concentration of vorticity. It is around this shear layer, as shown by the experimental observations, where the oscillation and fine grained turbulence developed, causing the flow to lose its organized structure and break up to full turbulence. A study of the nonlinear interaction and the energy transfer between different spanwise modes is also performed. The result of the study of the energy transfer between the spanwise modes for six different cases showed that, depending on the initial amplitudes and phase differences between the fundamental and harmonic modes, different evolutions are observed.

Avram Sidi, Technion - Israel, Institute of Technology, Haifa, Israel: "A Tutorial on Acceleration of Convergence of Iterative Methods with Applications"

The most common way of solving the large systems of linear or nonlinear equations that arise as a result of discretizing continuum problems (by finite differences or finite elements) is by iterative methods. In most cases of interest these methods suffer from poor convergence rates. Various acceleration of convergence methods to deal with this problem have been proposed and used recently. The topic of acceleration of convergence will be considered in a leisurely and informal manner. Convergence and stability results for various acceleration methods will be discussed and will be used in devising efficient strategies for implementing these methods. Applications to fluid mechanics problems will be mentioned. The acceleration methods that will be considered are algorithmically very simple and almost trivial to program. They can be implemented very efficiently on parallel machines.

Patrick Smolinski, University of Pittsburgh: "Parallel Solution of Structural Dynamics Problems on the Transputer System"

Several different parallel time integration algorithms have been evaluated for the analysis of transient structural problems. Explicit, explicit subcycling, and nodal partitioned implicit algorithms have been used to solve various size test problems. The problems were solved on a transputer system using different numbers of processors. Results indicate that a substantial speed-up can be achieved through the use of parallel processing.

Eli Turkel, Tel Aviv University: "A Seminar in Two Parts"

Part I. Runge-Kutta Central Difference Schemes. We will discuss the effect of artificial dissipation on central difference schemes. Changing the form of the viscosity or the boundary treatment can affect both the accuracy and the convergence rate of the scheme. Examples will be given for both 2D and 3D inviscid and viscous flows.

Part II. Asynchronous Difference Schemes and Parallel Computers. In parallel computers communication between the processors is a major chore of the operating system. We discuss algorithms for hyperbolic and elliptic partial differential equations that minimize communi- cation diffuculties. Stochastic models are introduced to simulate a multi-user parallel computer.

J. D. A. Walker, Lehigh University: "Mechanisms of Turbulence Production Near a Wall"

A central feature of turbulence production near the wall is associated with the process of the ejection of vorticity from the near wall region into the outer part of the flow in discrete events usually referred to as bursting. Calculation of such events is difficult owing to the violent unsteady nature of the phenomena. A factored algorithm using upwind-downwind differencing in Lagrangian variables is described which adequately resolves the onset of such eruptions. Recent experimental studies on the dynamics of a single hairpin-like vortex are discussed; these indicate that the hairpin vortex is a basic building block in boundary-layer turbulence and induces the eruptive effects commonly observed.

Yau Shu Wong, University of Alberta: "Absorbing Boundary Conditions for Hyperbolic Equations"

When computing a numerical solution to an unsteady partial differential equation on an infinite domain, it is necessary to perform the calculation on a finite computational domain. A uniform approach to construct absorbing boundary conditions for second-order hyperbolic equations is described. The nonlocal boundary condition, given by a pseudo-differential operator that annihilates travelling waves, is obtained through the dispersion relation of the differential equation by requiring that the initial boundary value problem admit only wave solutions with positive group velocities. Local approximation of this global boundary condition yields an $n-^{th}$ order differential operators. The effectiveness of the proposed boundary condition is demonstrated by computational examples.

WORKSHOPS

An avowed objective of the Case/NASA Lewis Institute for Computational Mechanics in Propulsion (ICOMP) is to develop computational capability in dealing with problems of fluid-solid interactions as they are found, for example, in turbomachinery. Accordingly a one day workshop on "Unsteady Phenomena in Turbomachinery" was held on July 20, 1988. The purpose of this was to bring together in an informal setting those ICOMP and NASA Lewis research personnel interested in the subject topic.

As can be seen from the program (Figure 6), talks were given by both Lewis and ICOMP personnel, and included presentations on experimental studies of fluid and aeroelastic effects in rotating machinery. The time slots allocated to each of the papers were such as to allow adequate time for discussion of each paper. The setting in the auditorium of the NASA Lewis Administration Building was comfortable and effective.

The morning program consisted of surveys of the larger picture, both computational and experimental, presented by Lewis personnel. The afternoon sessions in a very complementary manner highlighted the treatment of specific computational issues primarily by ICOMP personnel. The workshop was successful in making both Lewis and ICOMP people aware of the breadth of the ongoing activity in the subject area.

We are all very saddened by the collapse and eventual death of Dr. Krishna R. V. Kaza following his eloquent lecture on means for unifying fluid and structural formulations toward effecting proper computational solutions of fluid-structure interaction problems in turbomachinery.

A second one day workshop on "Dealing with Large Gradients in Computational Fluid and Structural Mechanics" was held at the NASA Lewis Research Center on Tuesday, August 16, 1988. This workshop was held because the problem of dealing with large gradients is pervasive in both fluid mechanics and structural mechanics computations. It was considered extremely useful to examine the needs and methods of handling large gradients, and to explore the applicability of techniques devised in either fluid or structural mechanics to the other field.

The workshop program, shown in figure 7, centered around 11 talks, with discussion, and a summary discussion session. The speakers were from NASA Lewis and from six universities; all the speakers had ICOMP association.

The topical material discussed included large gradients associated with shock waves, vortical structures, material interfaces, and cracks in structures, and discussions of a number of computational methodologies. Perhaps the most striking feature was the variety of methods discussed, both in the talks concerning applications and those discussing general methods.

OTHER ACTIVITIES

The ICOMP Advisory Committee held its second annual meeting on December 9, 1988. This committee consists of the following members:

Dr. Stanley G. Rubin, University of Cincinnati;
Dr. J. Tinsley Oden, University of Texas, Austin;
Dr. Earll Murman, Massachusetts Institute of Technology;
Dr. Ted B. Belytschko, Northwestern University; and
Dr. Steven A. Orszag, Princeton University

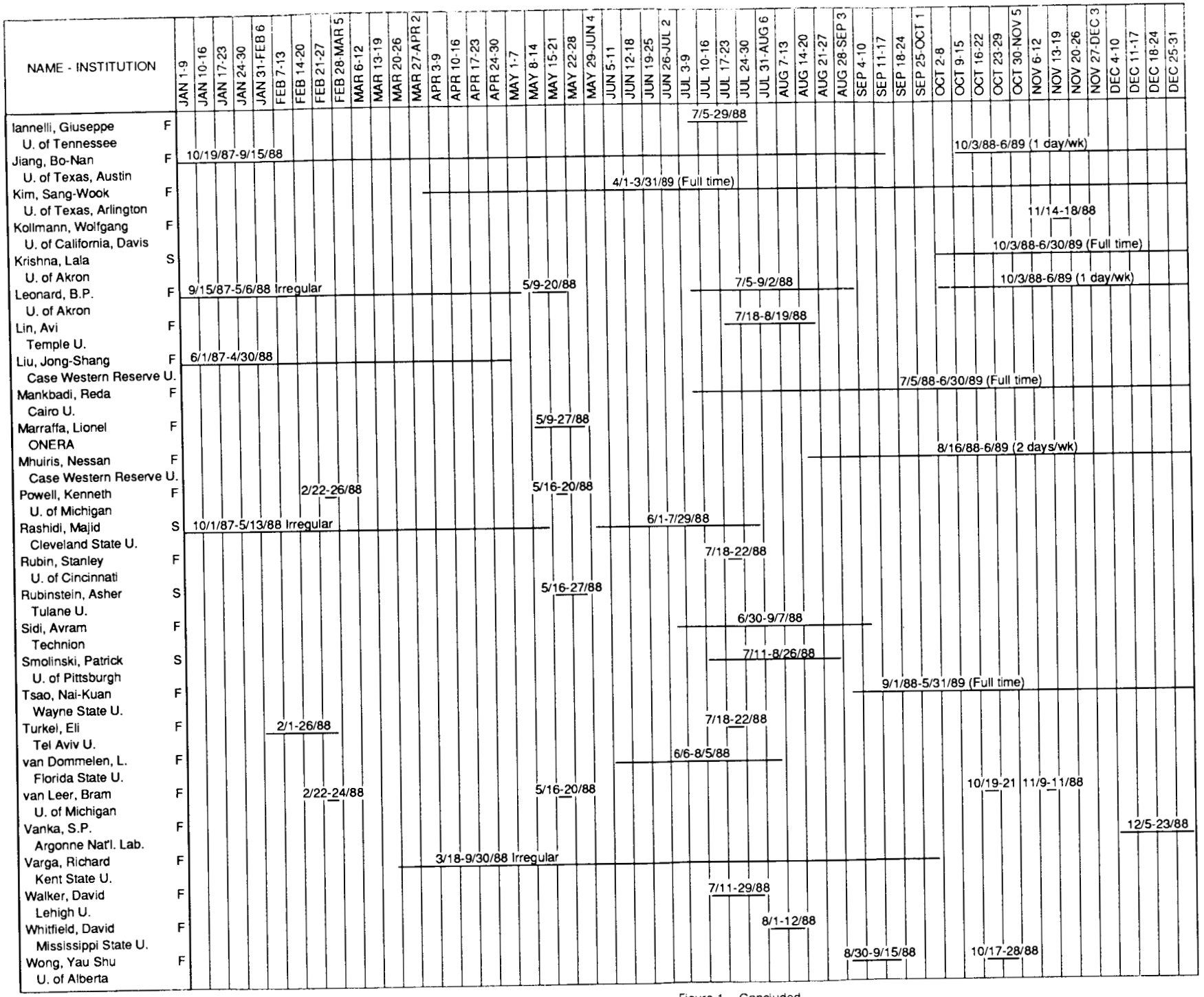
is charged with reviewing the ICOMP program and making recommendations on ICOMP activities. The selection of a Director for ICOMP was a major topic of discussion by both the ICOMP Steering and Advisory Committees at the 1987 meeting. Thus the announcement of Dr. John J. H. Miller of the Department of Mathematics at Trinity College in Dublin, Ireland was enthusiastically accepted at the 1988 meeting. Dr. Miller's selection was the culmination of a search of almost two years. He is expected to arrive at ICOMP in late summer of 1989 but will become involved in early 1989. ICOMP is anticipating continued growth under this leadership.

ORIGINAL PAGE IS
OF POOR QUALITY

S - STRUCTURES
F - FLUID MECHANICS

Figure 1. - ICOMP visiting researchers, 1988.

ORIGINAL PAGE IS
OF POOR QUALITY



S - STRUCTURES
F - FLUID MECHANICS

Figure 1. - Concluded.

ORIGINAL PAGE IS
OF POOR QUALITY

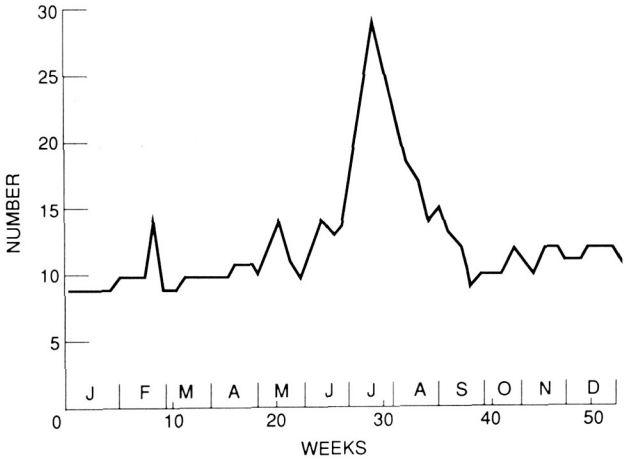


Figure 2. - ICOMP weekly resident staff, 1988.

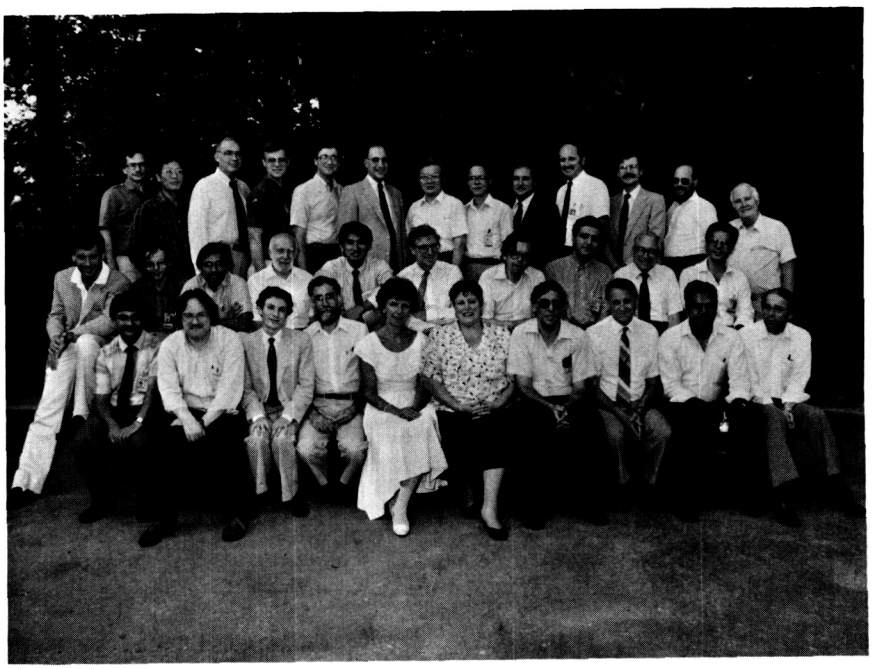


Figure 3. - ICOMP Steering Committee and visiting researchers in July, 1988. 1st row seated (left to right): Jitesh Gajjar, Lennart Hultgren, Joseph Iannelli, Avram Sidi, Christine Calvert, Darleen Midkiff, Avi Lin, Lou Povinelli, Suresh Aggarwal, Marvin Goldstein. 2nd row: Alberto Boretti, Leon Van Dommelen, Ammar Degani, Isaac Greber, Abhisak Chulya, Lester Nichols, Thomas Hagstrom, Rajid Rashidi, Eli Reshotko, Mohammad Farshchi. 3rd row: Alvin Bayliss, Sang-Wook Kim, B.P. Leonard, David Walker, Edward Bogucz, Stanley Rubin, Fred Chang, Bo-Nan Jiang, Robert Mullen, Brent Miller, Thomas Balsa, Eli Turkel, Charles Feiler.

ORIGINAL PAGE IS
OF POOR QUALITY

	UNIVERSITY OR INSTITUTION	NUMBER
1	ALBERTA	1
2	AKRON	3
3	CAIRO	1
4	CALIFORNIA-DAVIS	1
5	CAMBRIDGE	1
6	CASE WESTERN RESERVE	2
7	CINCINNATI	1
8	CLEVELAND STATE	3
9	COLUMBIA	1
10	EXETER	1
11	FLORIDA STATE	1
12	GEORGE WASHINGTON	1
13	ILLINOIS-CHICAGO	1
14	IMPERIAL COLLEGE	1
15	KENT STATE	1
16	LAGOS	1
17	LEHIGH	2
18	MANCHESTER	1
19	MICHIGAN	2
20	MISSISSIPPI STATE	1
21	NORTHWESTERN	1
22	OHIO	1
23	OHIO STATE	1
24	PENNSYLVANIA STATE	1
25	PITTSBURGH	1
26	PURDUE	1
27	STANFORD	1
28	SUNY-BUFFALO	1
29	SUNY-STONY BROOK	1
30	SYRACUSE	1
31	TECHNION-ISRAEL INSTITUTE OF TECHNOLOGY	1
32	TEL AVIV	1
33	TEMPLE	1
34	TENNESSEE	1
35	TEXAS-ARLINGTON	1
36	TEXAS-AUSTIN	1
37	TEXAS-SAN ANTONIO	1
38	TULANE	1
39	VPI & SU	1
40	WAYNE STATE	1
41	ARGONNE NATIONAL LAB	1
42	NIELSEN ENGINEERING	1
43	ONERA	1
		50

Figure 4. - Composition of 1988 ICOMP staff - organizations represented.

	1986	1987	1988
PEOPLE	23	43	50
SEMINARS	10	27	39
REPORTS	2	9	22
WORKSHOPS	1	0	2
PRESENTATIONS	7	0	21
UNIVERSITIES OR ORGANIZATIONS	20	28	43

Figure 5. - ICOMP statistics (1986 to 1988).

INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION

ICOMP WORKSHOP
ON
UNSTEADY PHENOMENA IN TURBOMACHINERY
WEDNESDAY, JULY 20, 1988
AD BLDG. AUDITORIUM

AGENDA

8:00 a.m.	Opening Remarks Eli Reshotko, Chairman, ICOMP Steering Committee, CWRU
	SESSION I
	Chairman - Stanley G. Rubin, University Of Cincinnati
8:15 a.m.	Numerical Simulation of Multistage Flows in Turbomachinery John Adamczyk - Office of the Chief Scientist, NASA Lewis
8:50 a.m.	Experimental Fluid Mechanics in Turbomachinery Michael Hathaway - Internal Fluid Mechanics Division, USAARTA
9:25 a.m.	Experimental Aeroelastics Of Propfans Oral Mehmed - Structures Division, NASA Lewis
10:00 a.m.	COFFEE BREAK
10:30 a.m.	Fluid-Structure Interaction Problems in Turbomachinery Krishna Kaza - Structures Division, NASA Lewis
11:05 a.m.	Time-Accurate Solutions of Rotor Stator Interactions Philip Jorgenson - Internal Fluid Mechanics Division, NASA Lewis
11:45 a.m.	LUNCH
	SESSION II
	Chairman - Robert Mullen, ICOMP Steering Committee, CWRU
1:00 p.m.	Algorithmic Improvements in Rotor-Stator Codes David Whitfield - ICOMP, Mississipi State University
1:35 p.m.	Multiple-Time Scale Turbulence Model for Unsteady Turbulent Flows S. W. Kim - ICOMP
2:10 p.m.	Computation of Unsteady 3D Separation Leon Van Dommelen - ICOMP, Florida State University
2:45 p.m.	COFFEE BREAK
3:00 p.m.	Effects of Interstage Aerodynamic and Structural Coupling on Vibration of Mistuned Bladed Disks Dare Afolabi - ICOMP, Purdue University
3:35 p.m.	Transputer-Based Parallel Algorithm in Vibration Analysis Fred Akl - ICOMP, Ohio University & L. James Kiraly - Structures Division, NASA Lewis
4:10 p.m.	Numerical Propulsion Simulation John Szuch - Internal Fluid Mechanics Division, NASA Lewis
4:45 p.m.	Adjourn

Figure 6. - Agenda for ICOMP workshop on unsteady phenomena in turbomachinery.

INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION

ICOMP WORKSHOP
ON
DEALING WITH LARGE GRADIENTS
IN COMPUTATIONAL FLUID AND STRUCTURAL MECHANICS
TUESDAY, AUGUST 16, 1988
AD BLDG. AUDITORIUM

AGENDA

8:15 a.m.	Opening Remarks Isaac Greber, ICOMP Steering Committee, CWRU
	SESSION I
	Chairman - Lester D. Nichols, Chief Structures Division, NASA Lewis
8:30 a.m.	Solution of Steady State One-Dimensional Conservation Laws by Mathematical Programming - John Lavery, Internal Fluid Mechanics Division, NASA Lewis
9:05 a.m.	Spectral Elements and High Gradient Concentrations - Karol Korczak and R.A. Wessel, CWRU
9:40 a.m.	ULTRA-SHARP Schemes - B. P. Leonard, ICOMP, University of Akron
10:15 a.m.	COFFEE BREAK
10:45 a.m.	Adaptation Criteria for Vortical Flows - Kenneth Powell, ICOMP, University of Michigan
11:20 a.m.	High Resolution Flux Splitting Methods - Meng-Sing Liou, Internal Fluid Mechanics Divsion, NASA Lewis
12:00 Noon	LUNCH
	SESSION II
	Chairman - Earll Murman, ICOMP Advisory Committee, MIT
1:15 p.m.	Finite Elements with Embedded Localization Zones - Ted Belytschko, ICOMP Advisory Committee, Northwestern University
1:50 p.m.	Using Mixed Finite Element Methods to Solve Problems with Large Gradients - John J. H. Miller, Trinity College, Dublin ,Ireland
2:25 p.m.	Specialty Functions/Formulations for Local Stress Gradients in Mechanics - Nesrin Sarigul, Ohio State University
3:00 p.m.	COFFEE BREAK
3:30 p.m.	Elastostatics Problems for a Bimaterial Interface - Robert Ballarini, CWRU
4:05 p.m.	A New Valid Asymptotic Implicit Integration Algorithm for Stiff Differential Equations of Unified Viscoplastic Constitutive Models Abhisak Chulya, ICOMP
4:40 p.m.	Workshop Wrap-up - Lester Nichols and Earl Murman
5:00 p.m.	Adjourn

Figure 7. - Agenda for ICOMP workshop on dealing with large gradients in computational fluid and structural mechanics.

NASA
National Aeronautics and Space Administration

Report Documentation Page

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
NASA TM-101961 ICOMP-89-1		

4. Title and Subtitle	5. Report Date
Institute for Computational Mechanics in Propulsion (ICOMP) Third Annual Report—1988	March 1989
	6. Performing Organization Code
7. Author(s)	8. Performing Organization Report No. E-4656
	10. Work Unit No. 505-62-21
9. Performing Organization Name and Address National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191	11. Contract or Grant No.
	13. Type of Report and Period Covered Technical Memorandum
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, D.C. 20546-0001	14. Sponsoring Agency Code

15. Supplementary Notes

Report produced by the Steering Committee, Institute for Computational Mechanics in propulsion, NASA Lewis Research Center (work funded under Space Act Agreement C99066G). Space Act Monitor: Louis A. Povinelli.

16. Abstract

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated jointly by Case Western Reserve University and the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1988.

17. Key Words (Suggested by Author(s))	18. Distribution Statement
Numerical analysis Computer science Mathematics	Unclassified—Unlimited Subject Category 64

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No of pages	22. Price*
Unclassified	Unclassified	60	

NASA FORM 1626 OCT 86

*For sale by the National Technical Information Service, Springfield, Virginia 22161